DIGIXCT SOFTWARE USER MANUAL

DigiXCT v3.0 RevA **January 2012**

Revision history

Revision	Date	Modifications
A	January 2012	Creation

For any technical question please contact Digisens at

Digisens SA
17, allée du lac d'Aiguebelette
BP278-Savoie Technolac
73375 LE BOURGET DU LAC Cedex
France
Phone : +33 (0)479 658 917
Fax : +33 (0)479 250 899
Email : support@digisens.fr

General information

DIGISENS reserves the right to make changes and improvements to all the products mentioned in this document without prior notification.

Users must periodically check, by using appropriate comparative methods, the conformity of their software. If this is not done, DIGISENS cannot be held responsible.

Under no circumstances can DIGISENS be held responsible for loss of data, for financial losses or any other sort of special or accidental damages, be they directly or indirectly caused in any way whatsoever.

Contents

1 GENERAL PRESENTATION OF DIGIXCT 9

1.1 Introduction 9
1.2 Minimum configuration requirements........................... 9
1.3 Installation 10
1.4 Protection system.. 10
1.5 Diagnostic window.. 11

2 VOLUME VISUALIZATION: DIGIOBS MODULE 11

2.1 Presentation of the default GUI 12
2.2 Management of preferences...................................... 12
2.2.1 Computation tab 14
2.2.2 Interface tab... 15
2.2.3 Graphics tab 18
2.3 Loading a volume 19
2.4 Volume visualization.. 21
2.4.1 Multi-display visualization 21
2.4.2 Volume slab visualization.. 22
2.4.3 Display parameters 23
2.4.4 The color palette 26
2.4.5 Light sources 29
2.4.6 Object reflection... 30
2.5 Managing a volume 31
2.5.1 Display management 31

2.5.2 Volume alignment 33
2.6 Data management 36
2.6.1 Saving and exporting volumes 36
2.6.2 Exporting slices 37
2.6.3 Exporting videos 39
2.6.4 Screen dumps 40

3 TOMOGRAPHIC RECONSTRUCTION: DIGIR3D PLUG-IN 50

3.1 Introduction 50
3.2 Workflow definition 50
3.3 Hardware definition 51
3.3.1 [Mandatory] - Camera definition 51
3.3.2 [Optional] – Geometry calibration 55
3.3.3 [Mandatory] - Geometry definition 62
3.4 Input 64
3.4.1 [Mandatory] - Input definition 64
3.4.2 [Optional] – Projection pre-processing 67
3.4.3 [Optional] – Phase contrast pre-processing 72
3.4.4 [Optional] – Projection correction 73
3.5 X-ray CT system compatibility 76
DigiR3D is compatible with the following CT system manufacturers: 76
3.6 [Mandatory] - Output definition 77
3.6.1 Volume format and name 77
3.6.2 Volume dimensions to be reconstructed 78
3.6.3 Region of interest 80

3.6.4 Volume resolution ... 82
3.6.5 Cylindrical reconstruction constraint ... 82
3.6.6 Reconstruction grid orientation ... 84
3.7 Reconstruction [mandatory] ... 84
3.7.1 Filtered back projection ... 85
3.7.2 Iterative algorithm ... 87
3.7.3 Post-processing ... 93
3.8 Preview and Adjustment ... 97
3.8.1 Preview Settings ... 97
3.8.2 Adjust ... 101
3.8.3 Metal artifact reduction ... 104
3.9 Save, load and reconstruction ... 106

4 ANALYSIS OF CUTTING PLANES: DIGICUT PLUG-IN... 107

4.1 General presentation of DigiCUT ... 107
4.2 DigiCUT functionalities ... 108
4.2.1 Resolution ... 108
4.2.2 Contrast ... 108
4.2.3 Moving in the image ... 108
4.2.4 Other features ... 108
4.3 Specific slice extraction modules ... 119
4.3.1 Cut Plane ... 119
4.3.2 Cut Cylinder ... 119
4.3.3 Cut Cone ... 122
4.3.4 Cut Radial Plane ... 124

5 VISUALIZATION AND EXTRACTION OF SURFACES: DIGISRF PLUG-IN.. 126

5.1 General presentation of DigiSRF............................. 126
5.2 DigiSRF functionalities 126
5.2.1 Surface manager 126
5.2.2 Surface operations... 127
5.2.3 Surface operations... 128
5.2.4 Simultaneous visualization of multiple surfaces ... 138
5.3 Saving/ reloading.. 139

6 IMPORT OF TOMOGRAPHY SLICES: DIGISIM PLUG-IN 140

6.1 General presentation of DigiSIM.............................. 140
6.2 DigiSIM functionalities 140

7 AUTOMATION OF RECONSTRUCTION TASKS: BATCH MODE... 143

7.1 General presentation... 143
7.2 Batch mode functioning 143

1 General presentation of DigiXCT

1.1 Introduction

DigiXCT is a software solution belonging to Digisens Company which allows:

- Calibration of tomography systems
- Reconstruction in different geometry configurations
- Visualization of volumes and meshes
- Manipulation of a 3D cutting planes
- Analysis of density levels based on histogram
- Basic measurements

The software is made up of 2 main modules and processing plug-ins. The two main modules are:

DigiOBS : Visualization and 3D analysis software
DigiR3D : Reconstruction software
DigiXCT

Its plug-ins are:

CUT	: Analysis of cutting plane & measurements
SRF	: Extraction and mesh visualization
SIM	: Import of slices
Batch Mode	: Module for producing macros by command line

The modules and plug-ins are brought together in the same interface based on the DigiObs module.

1.2 Minimum configuration requirements

Minimum recommended configuration for a desktop computer:

- Double slot PCIe motherboard
- Intel quad core processor
- 12Gbits RAM minimum (two times the total amount of GPU RAM)
- Windows XP / Windows 7 - 64bits

- 10000 rpm hard disks, mounted as RAID0 drive
- NVIDIA Fermi board dedicated to computation (Tesla board)
- NVIDIA board for visualization

Minimum recommended configuration for a laptop computer:

- Intel Dual Core Processor
- 4Go RAM
- Windows XP / Windows 7 64bits
- High speed hard disk
- Quadro 2000M graphic board
- **NO IGP ENABLED**
- **OPTIMUS TECHNOLOGY DISABLED**

For efficient and optimized configuration, please contact Digisens technical team at support@digisens3d.com

1.3 Installation

The installation of DigiXCT begins from the execution of the file *DigiXCT_X_XX_X.exe* (where X_XX_X corresponds to the installer version tag). Once the installer has been started, let yourself be guided by the installation wizard.

For any difficulties in installing or launching the application, please contact Digisens technical support.

1.4 Protection system

The DigiXCT software suite is protected by a license file and a USB key which is programmed and supplied by Digisens. It must be inserted before starting DigiXCT and must remain so when it is running otherwise the application will shut down. The key is installed when it is used for the first time, this only needs to be confirmed by clicking on the `Next` key when the Windows dialogue box `“Hardware device detected”` appears.

Once the installation is finished, you must click on `Finish`. An information bubble appears by the menu bar Start Windows with the message `“Your hardware is installed and ready to use”`.

1.5 *Diagnostic window*

When the application is launched, if the graphic configuration is not adapted, a diagnostic window appears (Figure 1.1). It shows graphics card reference, OpenGL version that is present and OpenGL extensions necessary which have been found or not. When the software is used for the first time, you can choose to disable this information window for future start-ups.

In the event of display issues, this information should be given to the technical support.

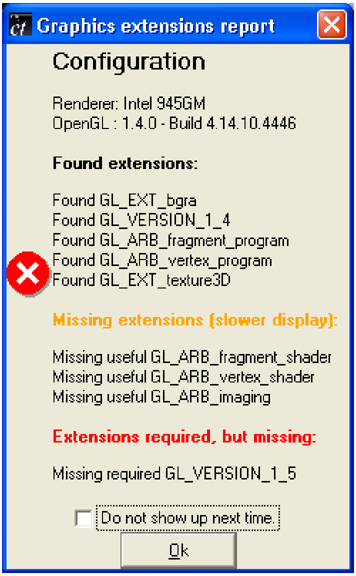


Figure 1.1: Diagnostic window

2 Volume visualization: DigiOBS Module

DigiOBS module corresponds to the main GUI of Digisens' product. It allows the visualization of datasets and provides all necessary links between the visualization and product features.

Maximum resolution depends on the capacity of your workstation. In this document, we are talking about a standard resolution of 512x512x512 voxels.

2.1 Presentation of the default GUI

DigiXCT appears in the form of a main window (Figure 2.1) made up of a 3D visualization area, a menu and a tool bar. The tool bar contains short cuts to the main functionalities and parameterization dialogue boxes.

2.2 Management of preferences

The advanced user has the opportunity to define his environment and a certain number of other parameters via the DigiObs preferences. To do so, you must open the preferences window which is found in the menu [`File → Preferences`] (Figure 2.2).

The preferences window displays 3 tabs:

- Computation : parameters useful for tomography reconstruction;
- Interface : user interface color parameters, management of windows and multi-display configuration;
- Graphics : display parameters.

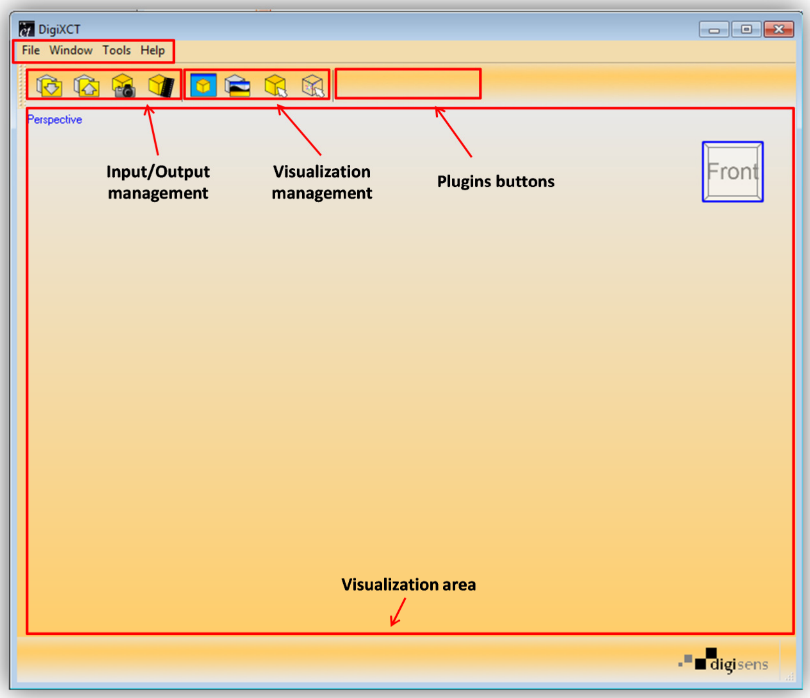


Figure 2.1: DigiXCT main window

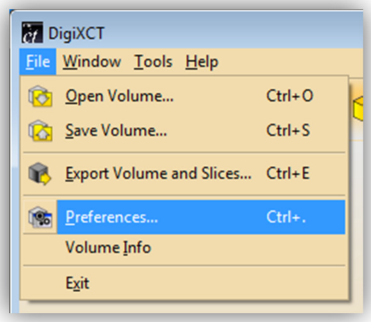


Figure 2.2: Opening of the preferences window

2.2.1 Computation tab

This tab has two fields used for reconstruction. These parameters are straightforwardly used by DigiR3D plug-in (`CPU Reconstruction` and `GPU Reconstruction`) (Figure 2.3).

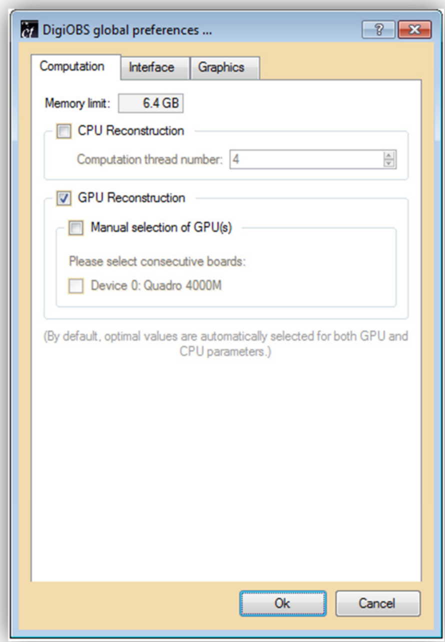
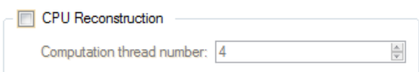
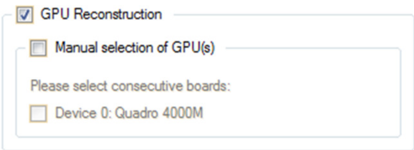


Figure 2.3: Preferences window computation tab



The number of threads for computation corresponds to the number of processes that can be launched in parallel onto the CPU. It depends on the workstation's architecture and more specifically on the total amount of cores in processors. The preferred memory size for the calculation of a sub-grid corresponds to the maximum memory that can be allocated for the calculation. This value depends on the amount of RAM memory available on the computer. It corresponds to the remaining memory once all the system processes have been launched.

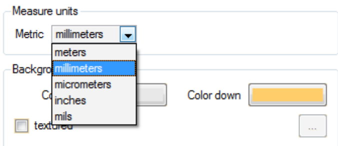


The `GPU Reconstruction` check box enables the use of GPU(s) for reconstruction.

This menu also allows the user to explicitly choose GPU board(s) used for reconstruction using SnapCT among a list of all available Cuda compatible devices.

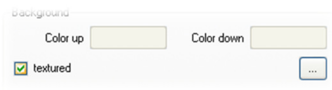
2.2.2 Interface tab

DigiXCT proposes a tool for informative measurements (see DigiCUT plugin). To be adapted to the system of units of one's country, it is possible to convert measures in mm (international system of units) or in imperial system of unit (ie inch, mils…).



Important note: This change in units is not applied to tomography geometry definition that remains in the international system of units.

The user can, by using this interface (Figure 2.4), change the background colors of DigiObs. He can either use a plain color by putting the same color in `Background color up` and `Background color down`, or create gradients.



`Textured` option allows the use of an image as wallpaper for DigiOBS. Compatible formats are jpg, bmp or png. This image replaces colors defined in `Color Up` and `Color Down`. The color `Selected color` corresponds to the color of the elements selected in the 3D grid and `Outline color` to the color of the physical limits of the visualized 3D volume.

The user can also change the size of icons by selecting `Icon size` among the 4 available formats.

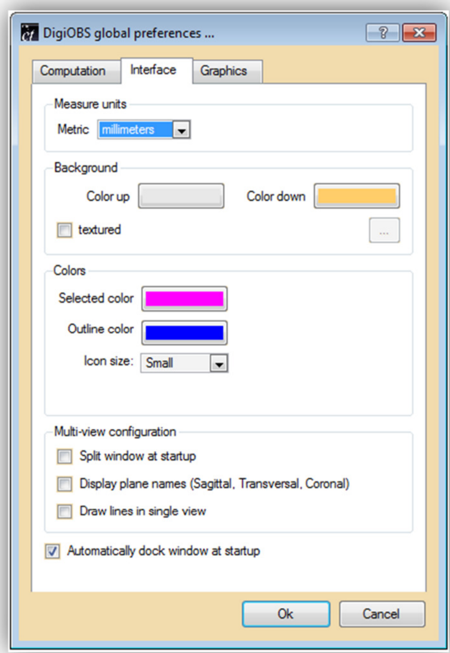


Figure 2.4: Preferences window interface tab

The `Multi-display configuration` allows the parameterization of multi-display as will be described in chapter 2.4. The first box allows the default activation upon launching DigiObs of a multi-view work window. The `Display plane names` box allows the X, Y, Z slices plane to be renamed in the GUI as Sagittal, Transversal and Coronal views. By default, these names are `Front`, `Side` and `Top`. `Draw lines in single display` allows the tracing of the edges of the planes intersecting the active display in the visualization.

Finally the box “`automatically dock window at startup`” allows the analysis window to be integrated into the main DigiXCT window as shown in figure 2.5.

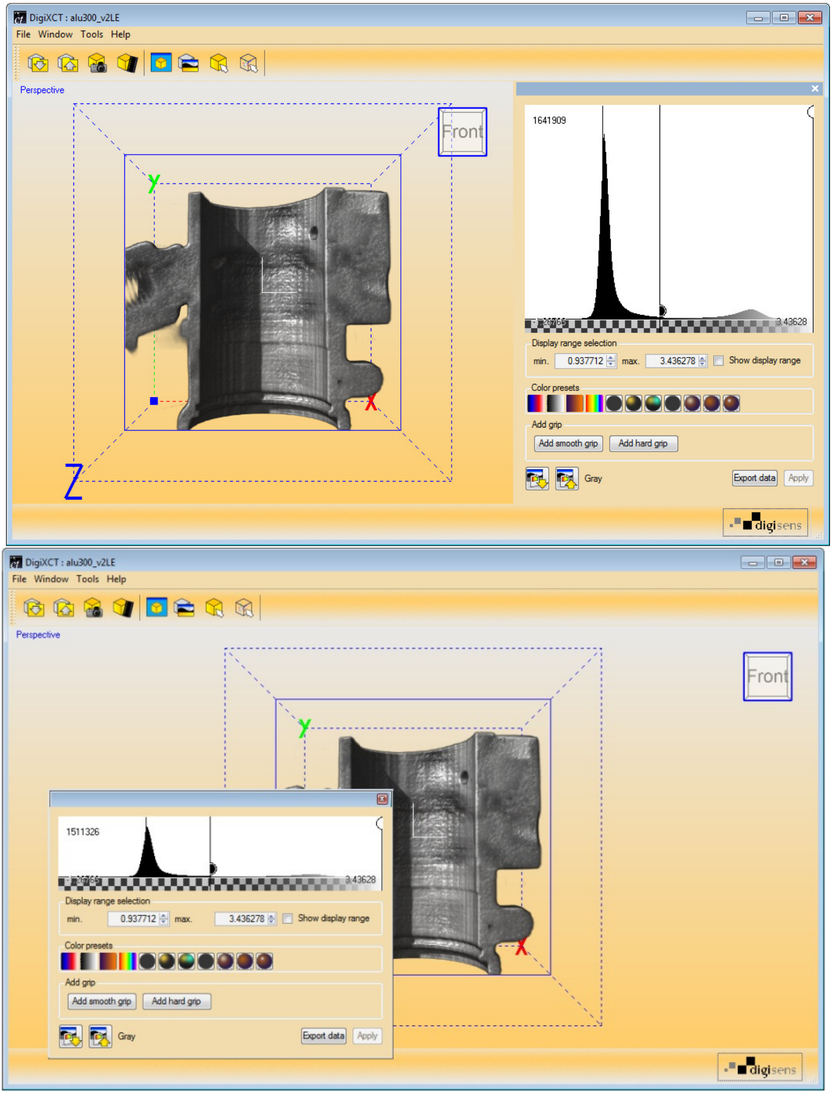


Figure 2.5: Top image: histogram window not docked with the interface. Bottom image: window docked with the interface.

Default configuration corresponds to docked windows. To get back free displaying of the windows, the user can either tick the box

`Automatically dock window at startup` in the preferences, or left-click on the window banner and still keeping the button pressed down, drag the window to the middle of the main DigiObs window.

To dock a window you only have to double click on the upper bar of the window or drag it to the edge of the main window.

2.2.3 Graphics tab

DigiOBS offers a powerful and realistic 3D rendering tool. To enable this visualization module, you only have to tick the `volume render` feature as described below.

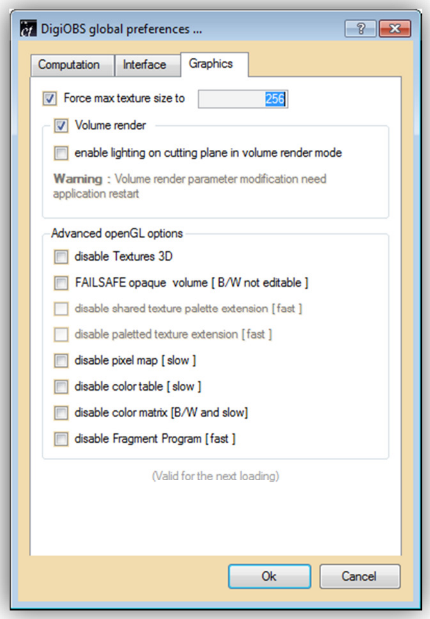


Figure 2.6 Preferences window graphics tab

The actions of each of the boxes (see figure 2.6) are as follows:

- FAILSAFE opaque volume: allows all of the openGL options to be invalidated. The final rendering is very poor. It is however

compatible with any type of graphics card.

- Disable volume render: deactivation of 3D rendering mode by ray tracing method. This mode is only recommended for graphics cards which have performance equal to or better than NVIDIA 8800/8600 cards.
- Fragment Program, shared texture and palette extension, palette texture extension, pixel map, color table, color matrix, texture 3D: increasingly slower and decreasingly powerful display mode.
- Force max texture size to: current graphics cards know how to manage 256 or 512 depending on the type of card.

2.3 Loading a volume

This action is possible either by using the command `Open Volume` of the menu [`File`], or by dragging down a file *.vol.

Dragging down a file consists of left-clicking on the file, holding down mouse button and then dragging the cursor into the application window, then releasing the button (Figure 2.7).

The loaded volume file's name appears in the window title (Figure 2.8). To manipulate the volume, you only need to use the mouse and the keyboard.

NOTE: For an optimal navigation in DigiOBS GUI, it is strongly advised to use a mouse with a roller wheel.

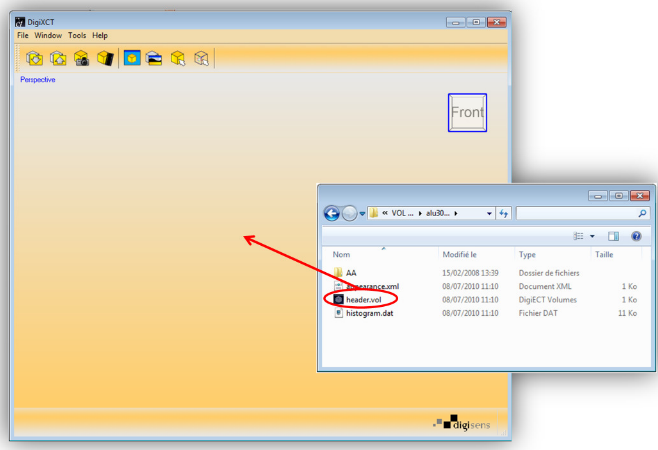


Figure 2.7: Opening a volume file

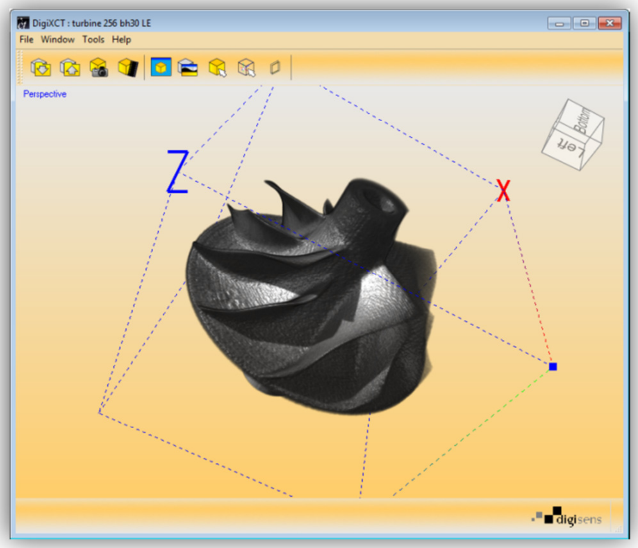


Figure 2.8: Visualization of turbine blades

2.4 *Volume visualization*

2.4.1 Multi-display visualization

DigiObs offers simultaneously the display of the 3D rendered object and 3 cutting planes.

Six display modes are available using the menu [Window] or using the button Switch Mode . These modes are:

- 3D display 
- 3x1 display 
- 2x2 display 
- Side display 

- Top display 

- Front display 

The Switch mode button allows the modes to be changed one by one.

Figure 2.9 shows an example of a 3x1 visualization.

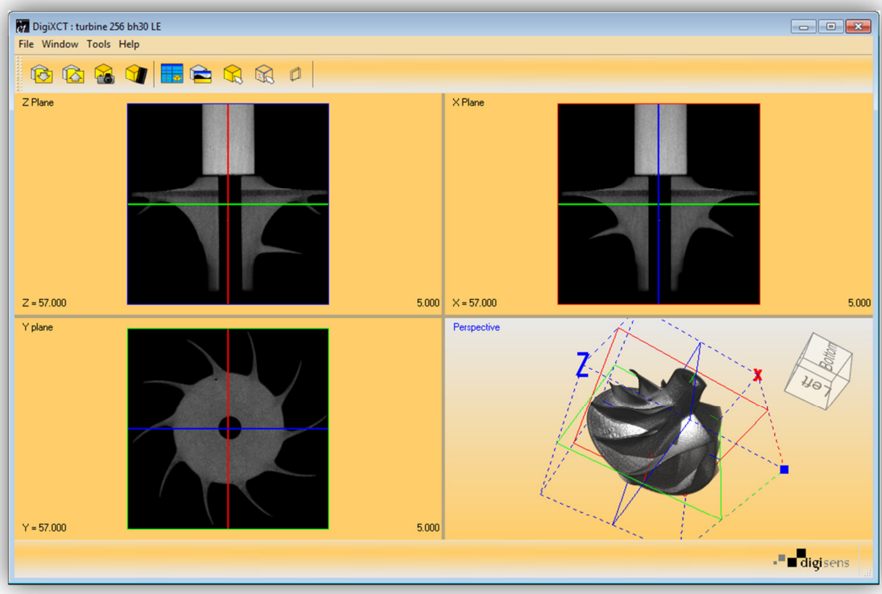


Figure 2.9: Management of the main window in 3x1 window mode

2.4.2 Volume slab visualization

This mode allows the display of a slab of the loaded 3D volume. This mode is only available with `volume render` activated (see preferences menu). To enable this mode, just click on the button . When the mode is active, the button becomes yellow .

Figure 2.10 shows how to change slab thickness using the dedicated parameter `Thickness` at the bottom of the GUI. Slab thickness is in millimeter.

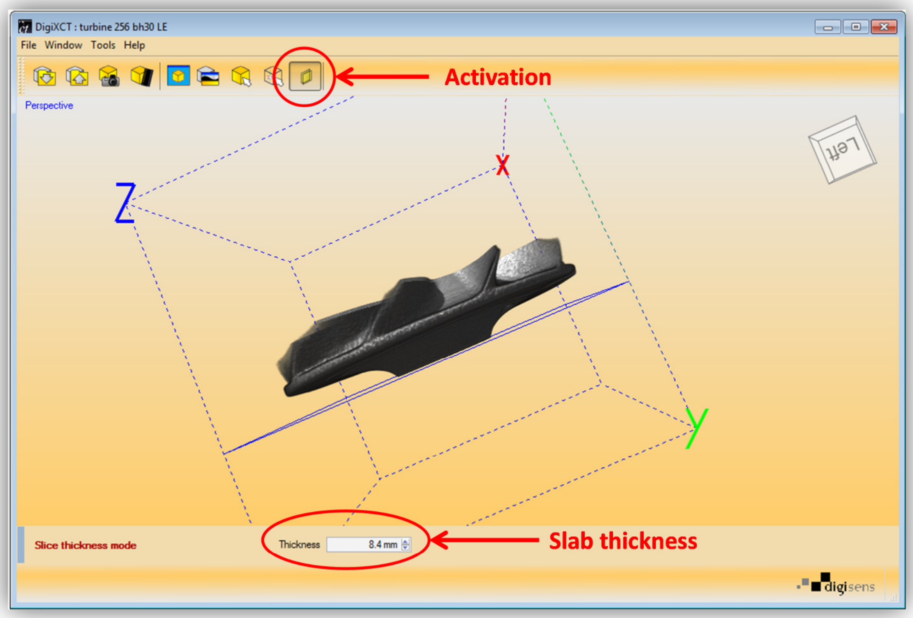


Figure 2.10: Volume slab display

Note: When this mode is un-adapted with other mode functionalities of DigiXCT, the activation button is grayed .

2.4.3 Display parameters

To visualize a volume, it is necessary to choose the range of density values that will be displayed.

It is possible that the visualization of a volume is subject to interference such as noise or un-adapted display LUT (Figure 2.11). In such a case, you must click on the "`Volume Appearance`" icon on the tool bar in

order to open the parameterization window. The diagram shown is the histogram of the object's density. It indicates in abscissa, the density values describing the volume, and as coordinates, the number of voxels (parts making up the volume) having these densities.

To modify the density range, you have to move the minimum and maximum limits around the zone symbolizing the object (either by moving the 'hard grip' in the diagram or by modifying the *min* and *max* values shown below the diagram) – see figure 2.12. It involves the second largest zone, the largest symbolizing the spike in the data curve. In order to refine this zone, you can zoom in and out between the min and max ranges by choosing the option "`Show display range`" (Figure 2.13). The volume appearance is updated automatically.

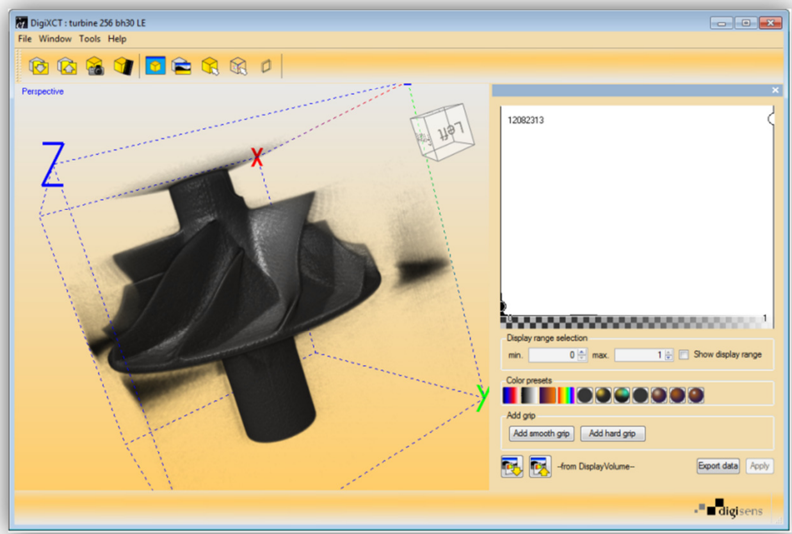


Figure 2.11: Wrong density range

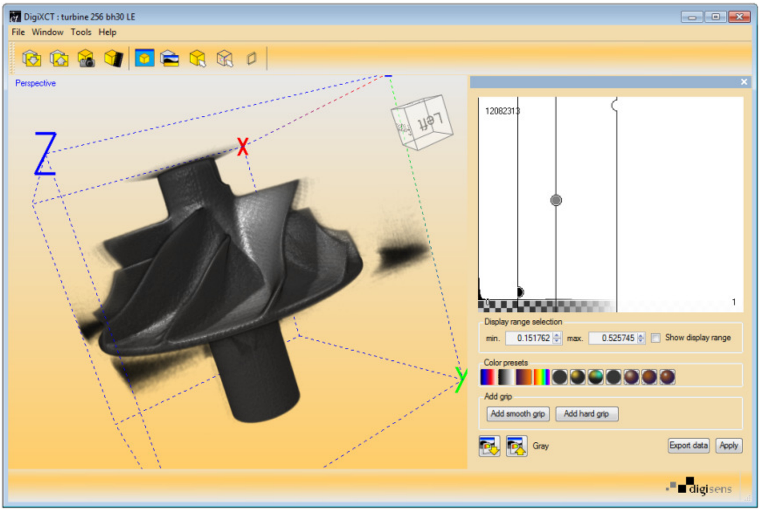


Figure 2.12: Parameterization window: choice of the density zone to be displayed

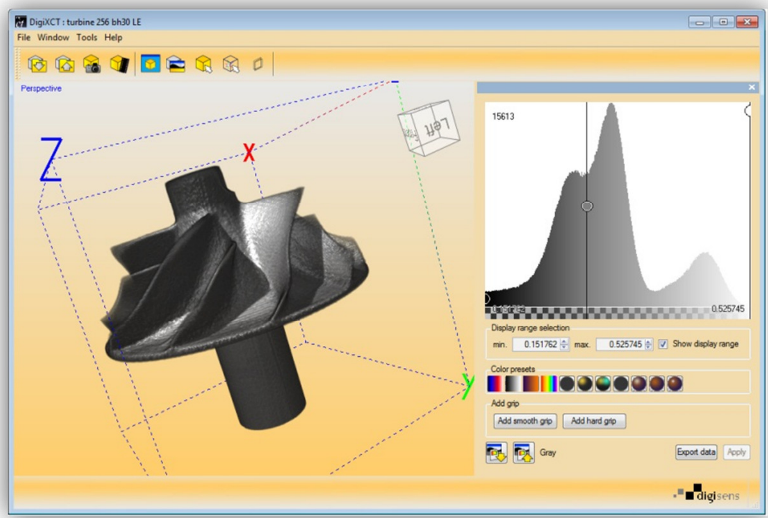


Figure 2.13: Focus in on the density range

A contextual menu in the histogram window allows data to be exported in

Excel format (right click on the histogram zone then select `Export data...`).

2.4.4 The color palette

The setting of color parameters is carried out with the palette located under the density profile in the window `Volume Display Parameters` (Figure 2.14). There are 5 pre-defined palettes (`Panel table presets`). The color palettes can be recorded and loaded via the `Load` and `Save` buttons. Figure 2.15 shows palette application on a specific example.

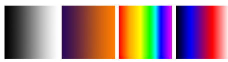


Figure 2.14: Pre-defined display colors

It is possible to add key points known as *grip.*
To do so, you must right click the mouse in a given place in the palette (figure 2.16), then select the command "`add`" from the menu which appears. To choose the color used at this *grip*, you must double click on the *grip*, and then select a color from the palette (Figure 2.17). It is possible to slide horizontally the grips using Alt+left mouse button.

Transparency adjustment is done directly using the little dot or the triangle placed on the grip and using Ctrl+left mouse button. Vertical movement upwards increases opacity and downwards vertical movement increases transparency. Display is updated automatically.

The `Apply` button allows validation of the palette and the min/max ranges chosen on the histogram. It triggers the update of data in the 3D visualization window.

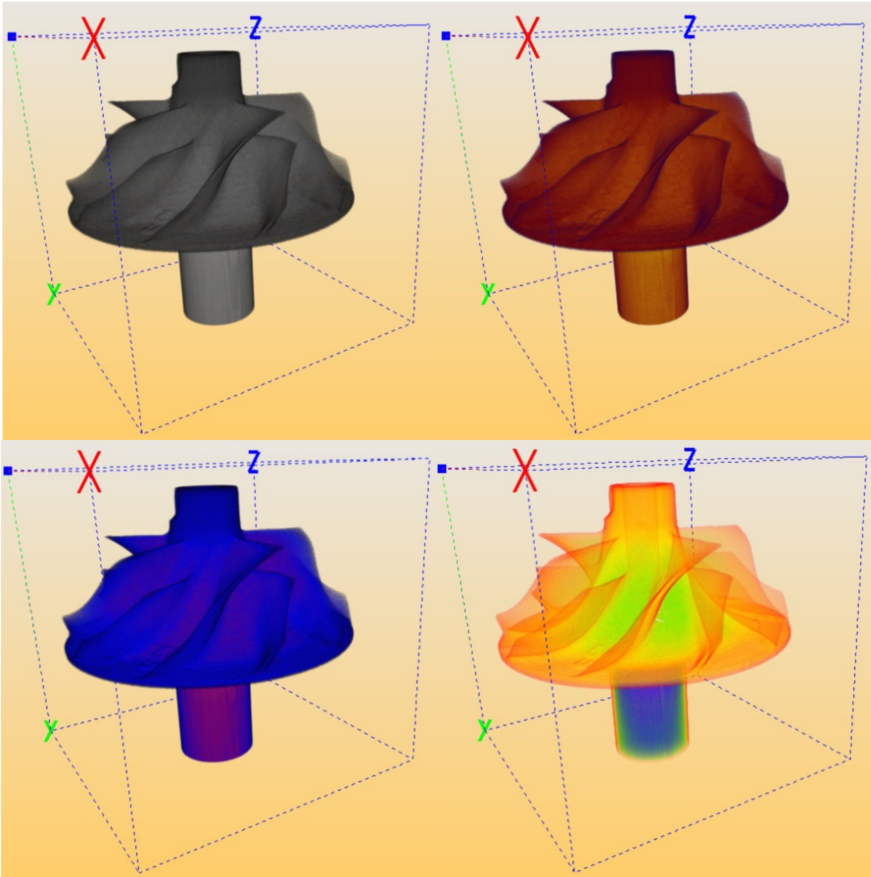


Figure 2.15: Visualization of the 3D part with pre-defined display colors: Grey Table, Heat Table, Rainbow Table and Blue Red Table.

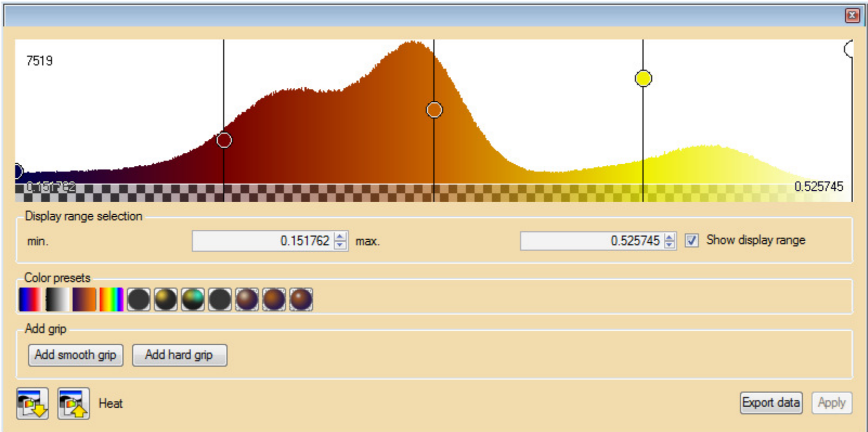


Figure 2.16: Density profile

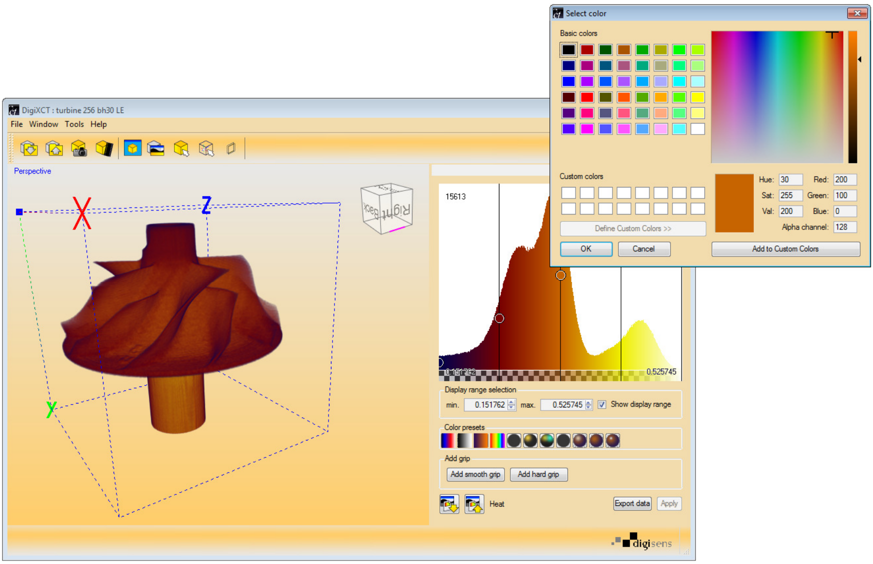


Figure 2.17: Color palette

2.4.5 Light sources

It is possible to add light sources via the 3 buttons shown in figure 2.18.

Figure 2.18: Control buttons for 3D scene illumination modes

The effects of these sources on the 3D object are shown in figure 2.19. Adding light sources allows playing with shadows at the same time as with saturation on the part. Illumination allows the user to analyze the 3D structure of their part in a more refined way.

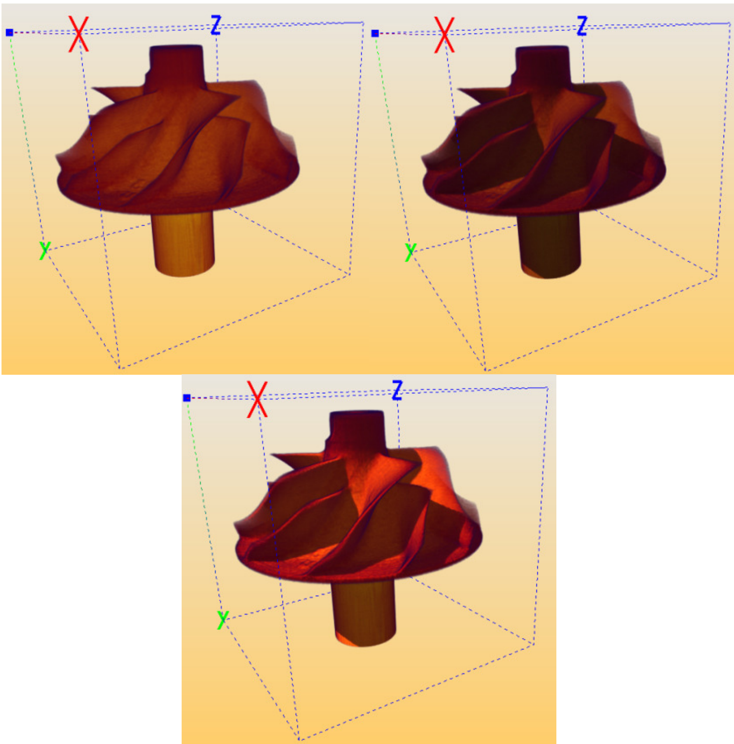


Figure 2.19: Example of applying light source management

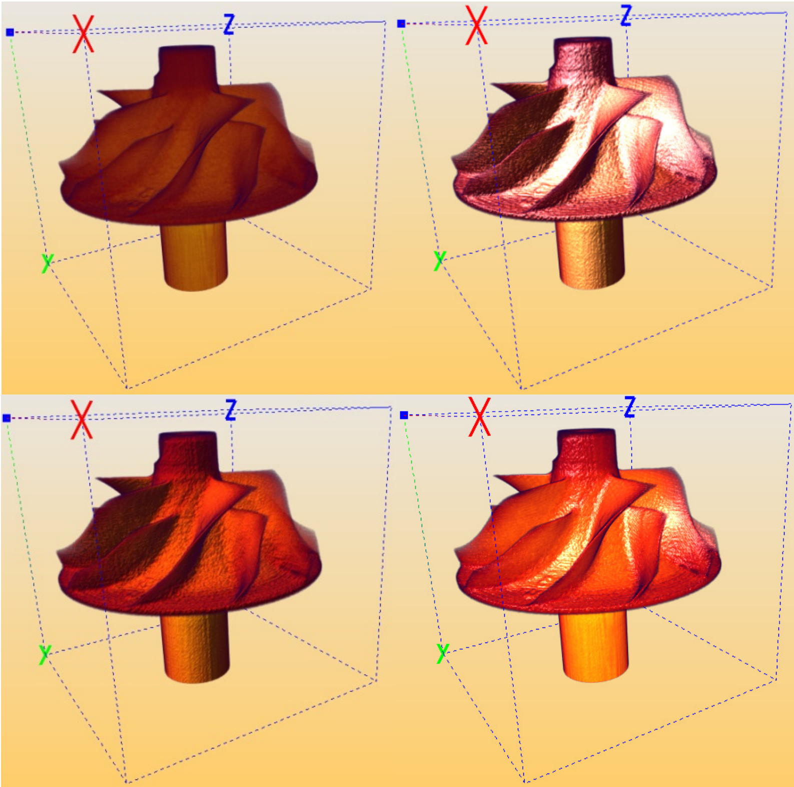


Figure 2.20: Example of applying reflection management

2.4.6 Object reflection

It is possible to adjust the amount of reflection on the objects. It is also possible to have a more metallic rendering of some parts than others. This management of reflection is achieved by using 4 buttons shown in figure 2.21. Hence, the user can display a mat finish, a plastic type finish, a metallic finish or a mirror finish.

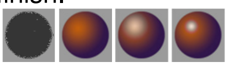


Figure 2.21: Control buttons for reflection modes

The effects of these sources on the 3D object are shown in figure 2.20.

2.5 *Managing a volume*

2.5.1 Display management

Using the mouse:

It is possible to navigate around the volume object with the help of the mouse (see figure 2.22). There are two modes of moving: one mode for moving the volume and one mode for moving the cutting plane. You can switch between these two modes by using the "`Mouse control mode`" button of the tool bar or by depressing the spacebar on the keyboard.

The icon "`3D volume`"  becomes "`cutting plane`" mode 
The cutting plane is moved by manipulating the mouse in the same manner as for moving the volume. An example of a cutting plane is shown in figure 2.23.

The left button of the mouse allows the cutting plane to be turned. The right button allows translation in the plane, and the roller wheel allows perpendicular translation to the plane. Keyboard key strokes can also be used and are shown in figure 2.24.

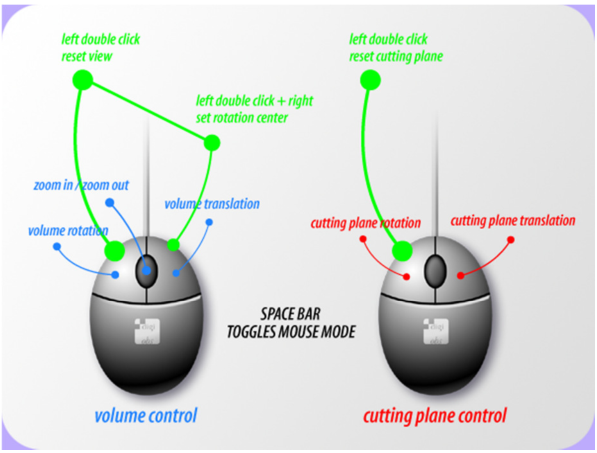


Figure 2.22: Use of the mouse for the manipulation of 3D volumes

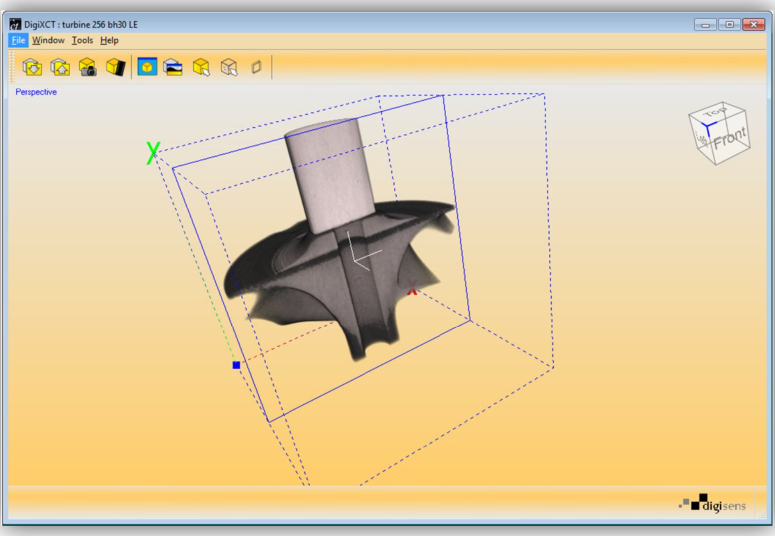


Figure 2.23: Use of cutting plane

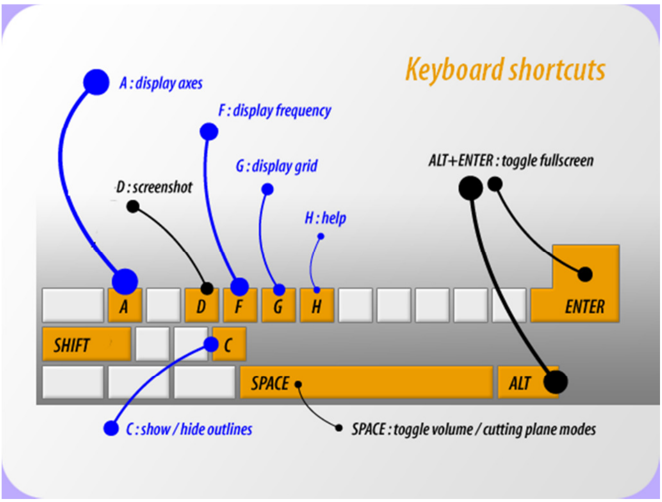


Figure 2.24: Details on active key strokes

2.5.2 Volume alignment

Displayed volumes are expressed in a direct reference axis which center coincides with the center of rotation of the system. This reference axis, called world reference (figure 2.25) can be activated with the keyboard shortcut "a". It is used to determine the center and the orientation of a section extracted from the volume.

DigiOBS uses two other reference frames:

- A reference frame in relation with the region of interest described in the chapter about DigiR3D and named volume reference (figure 2.25)
- A reference frame for measures allowing to express them regarding to an alignment that can be in relation with CAD model (measure reference in figure 2.25)

It is thus possible to align the volume by moving and/or rotating the measure reference. This feature is very useful in measure mode because it will allow expressing the location of an entity with regard to an absolute reference which can be a CAD reference for example.

To enable this mode, click on the button . When the mode is active, the button becomes yellow  and fields appear at the bottom of the GUI (figure 2.26).

Note: This mode may be un-adapted with other mode functionalities of DigiXCT. In this case, the button is grayed .

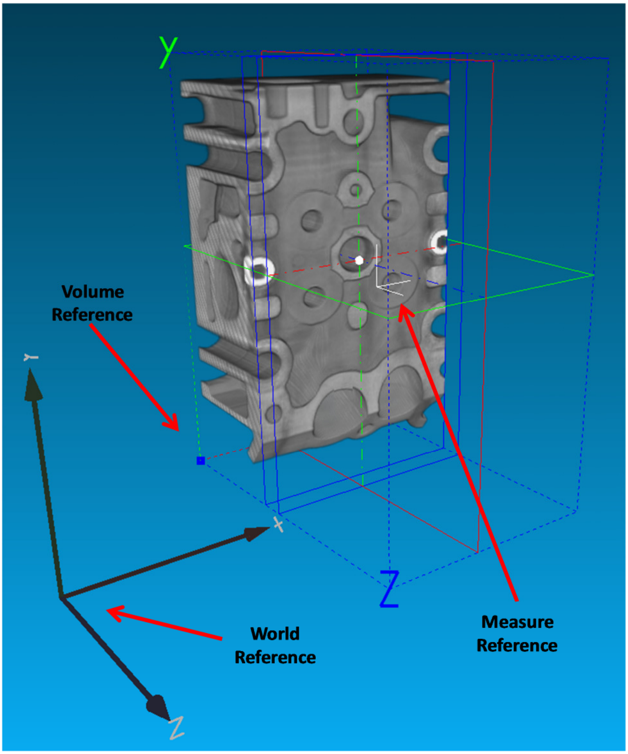


Figure 2.25: Reference axis definition in DigiXCT

During alignment process, the former orientation of the volume remains displayed with dashed lines. The new reference is defined by:

- Cutting planes axis red, green and blue
- Intersection of those three planes

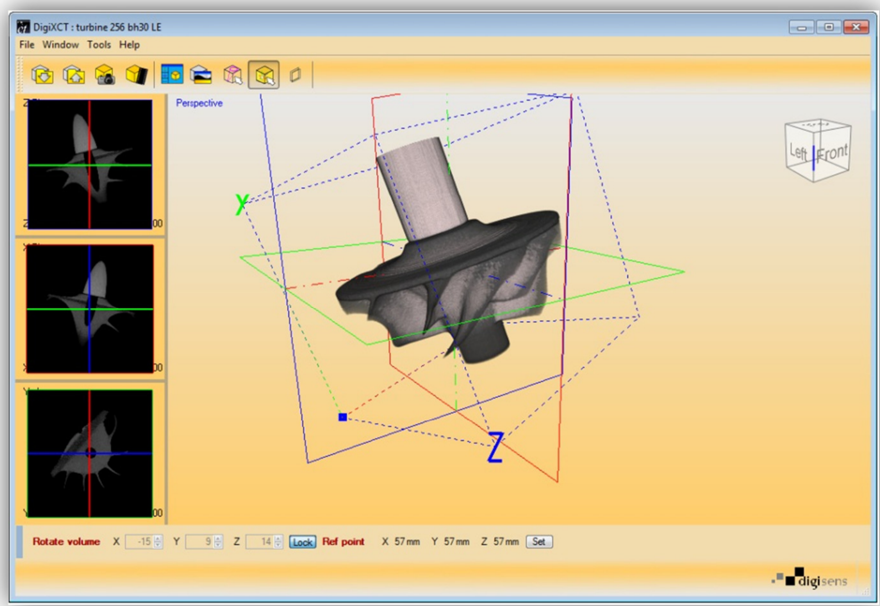


Figure 2.26: Volume alignment

To fix the orientation of the volume, just click the button `Lock` at the bottom of GUI and move the volume. Once the orientation is fixed, click again on `Lock` to validate the orientation.

To fix the origin of the new reference axis, just locate the origin by navigating in the three 2D views on the left of the GUI in figure 2.26 and click on `Set` button. The origin corresponds to the intersection of the three planes.

When everything is finished, all measures described in DigiCUT chapter will be expressed regarding to this reference axis set.

2.6 Data management

2.6.1 Saving and exporting volumes

Saving volumes is carried out thru the menu `Save Volume` of the menu [`File`]. It can be directly enabled by the icon .

Saving occurs on the currently displayed volume (dimensions, resolution, display LUT …). The available formats are a proprietary format (*.vol) and two other formats to provide compatibility with third part analysis software (Visilog im6 and Volume Graphics vgi formats).

To save the volumes in another configuration of dimensions, the user must use the menu `Export volume` of the menu [`File`].

This menu allows the displayed volume to be exported thru modification of some of its parameters:

- Volume dimensions (Figure 2.27): by ticking `Resize`, it is possible to choose the final dimension of the output volume

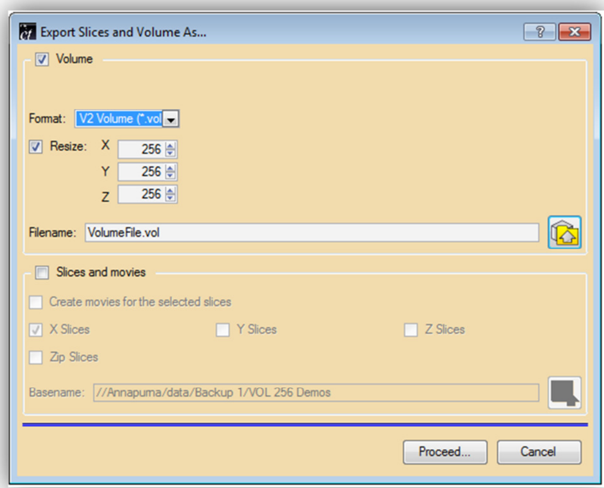


Figure 2.27: Modification of data dimensions

- Output volume format (Figure 2.28): The volume can be written in the current format of the volume (v2), either in a format compatible with Volume Graphics (VGMax or VGStudio) software or the Visilog software.

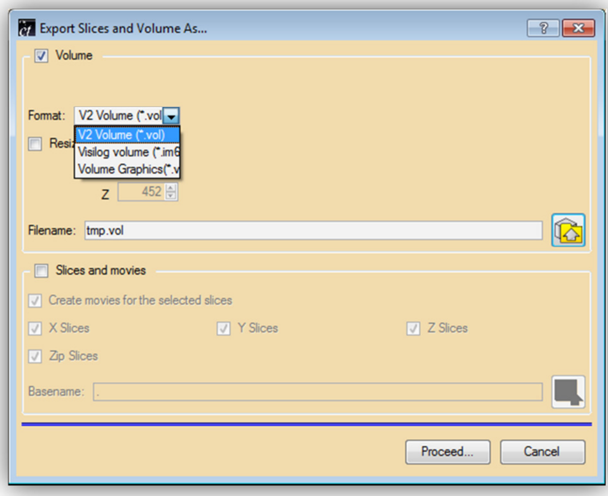


Figure 2.28: Export formats

2.6.2 Exporting slices

Files containing the description of a volume have the extension *.vol. The DigiObs module allows the visualization of such a volume in 3D with reduced resolution. It is possible to extract a series of slices in different file formats (bmp, tif, raw, jpeg). Slices can be exported according to the X,Y,Z orientations. For custom orientations, see chapter on DigiCUT plug-in.

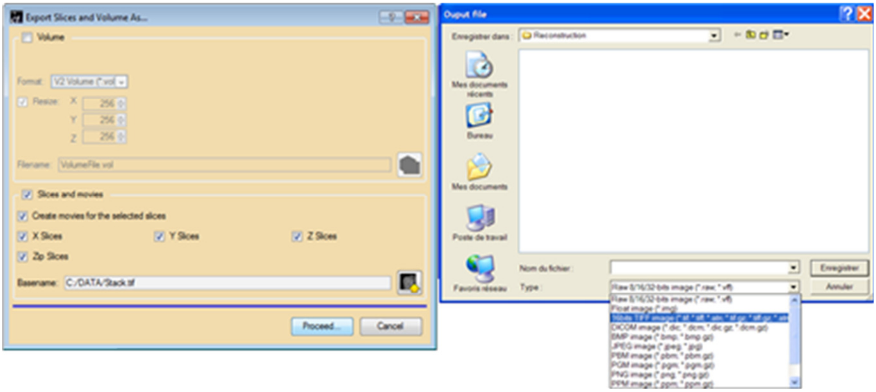


Figure 2.29: Slice extraction dialogue box

You must choose `[File → Export Volume]` in the menu. Then tick `Export Slices` (Figure 2.29).

To carry out a slice extraction, in the Base name you must specify a destination directory and a root for the name of the extracted images. If you choose c:\object\slices\slices_,in the c:\object\slices, directory you will get a series of images called slices_0001.bmp,slices_002.bmp, …. It is also necessary to specify the direction of the slices (X, Y, Z) and whether an individual compression of the images is required or not. For custom slices, the user will use DigiCUT plugin.

The slice export is carried out with the visualization parameters previously chosen in 3D in gray levels.

In the export directory concerned, the program automatically records an *.html file which pulls together in a table all of the data concerning the volume and the tomographic slices. See below an example of contents generated during the export of 1024 slices 1024x1024 pixels in the direction of the X axis.

X axis export

number of slices	1024 slices
image resolution	1024 x 1024 pixels
image size	2 cm x 2 cm
pixel size	19.53 µm x 19.53 µm

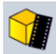
Volume information

Resolution	1024 x 1024 x 1024 voxels
Size	width 20 mm , height 20 mm , depth 10 mm
Voxel size	19.53 x 19.53 x 19.53 µm
Expanded space required (memory)	4.1 GB
Compressed space required (disk)	1.6 GB
Disk-storage in 17 subgrid(s)	- basename C: /Reconstruction/Object.vol - slices per sub-grid [65 65 65 65 65 65 65 65 65 65 65 65 65 65 65 65 60]
Display range	[0 : 0.95]

It is possible to make a video which allows the slices to be displayed one after the other. To do this, you only have to tick the `Create movies for the selected slices`.

2.6.3 Exporting videos

The command `[Window → Export Movie]` allows a video to be exported in the 3D visualization zone, either in AVI format ("`save images`" box not ticked) or in the form of a series of images (see figure 2.30). You only need to specify the name of the AVI file or the base name of the images in the corresponding field. You then have to click on the "`Record`" button to start the recording and then on the "`Stop`" button to stop it. A new window offers a choice of compression codec available in your computer registry directory. The film thus produced is the output of the 3D zone and of all the events that influenced it during the recording.

This function can also be activated via the button  .

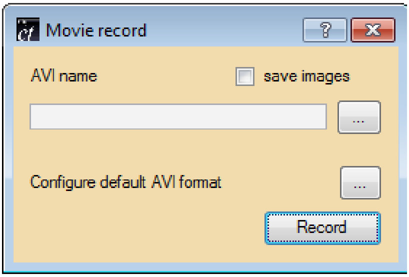


Figure 2.30: Film export dialogue box

Note: It is possible to remove cube borders in the 3D visualization area (symbolizing the inclusive volume) or to display it with pressing the ‘c’ key.

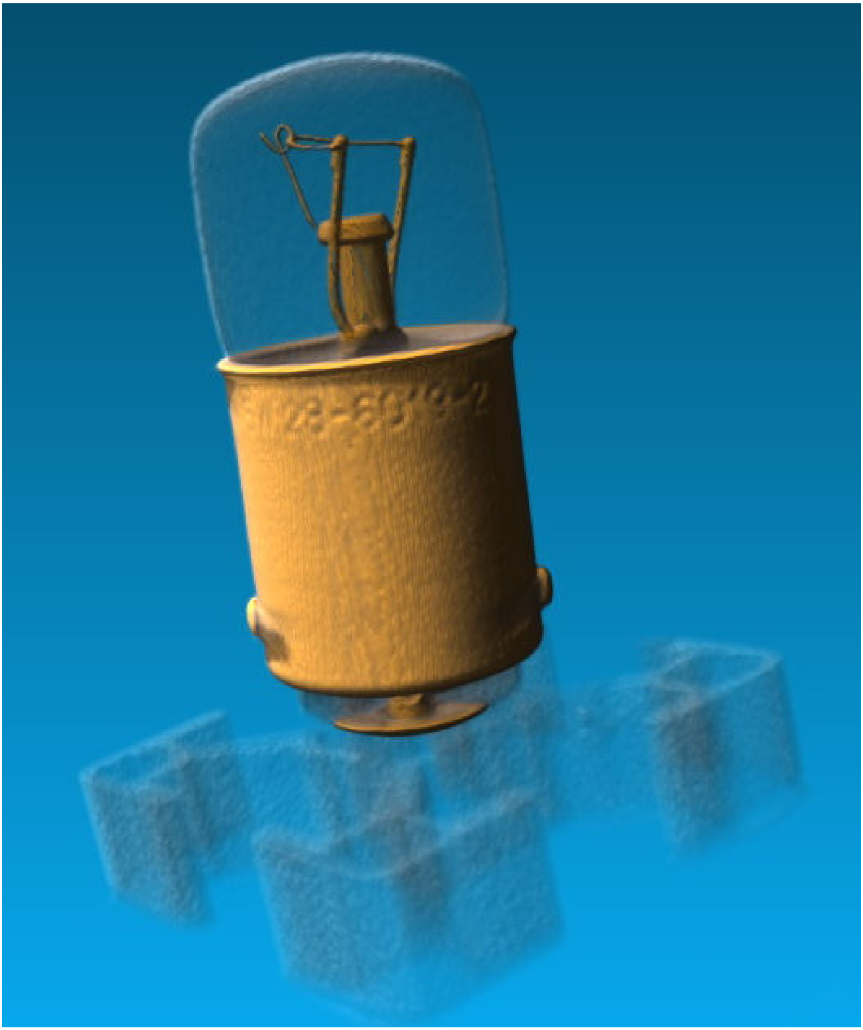
It is possible to record a high resolution animation. The principle is quite simple: the function allows you to compute the entire trajectory of a camera moving around or inside the object from discrete camera locations. To do so, some keyboard keys have to be used:

- Alt+F1 : save the actual camera location (a message appears bottom left saying “Position 1 saved”)
- Alt+F1 twice : erase the entire path saved
- F1 : play/stop the animation

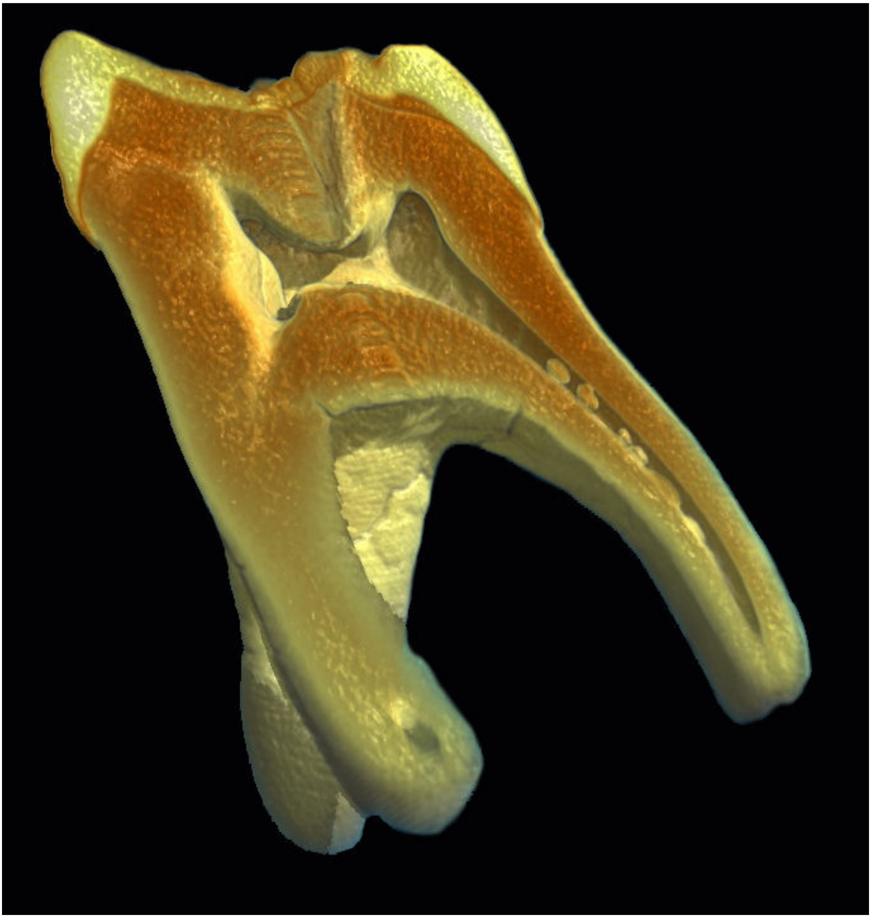
2.6.4 Screen dumps

The command `[Window → Export Image]` allows a screen dump of the current display to be carried out. Once the mode has been activated, you only have to choose the directory, the name of the file to be saved as well as the file format (jpg, png, eps, ps, ppm, bmp and fig).

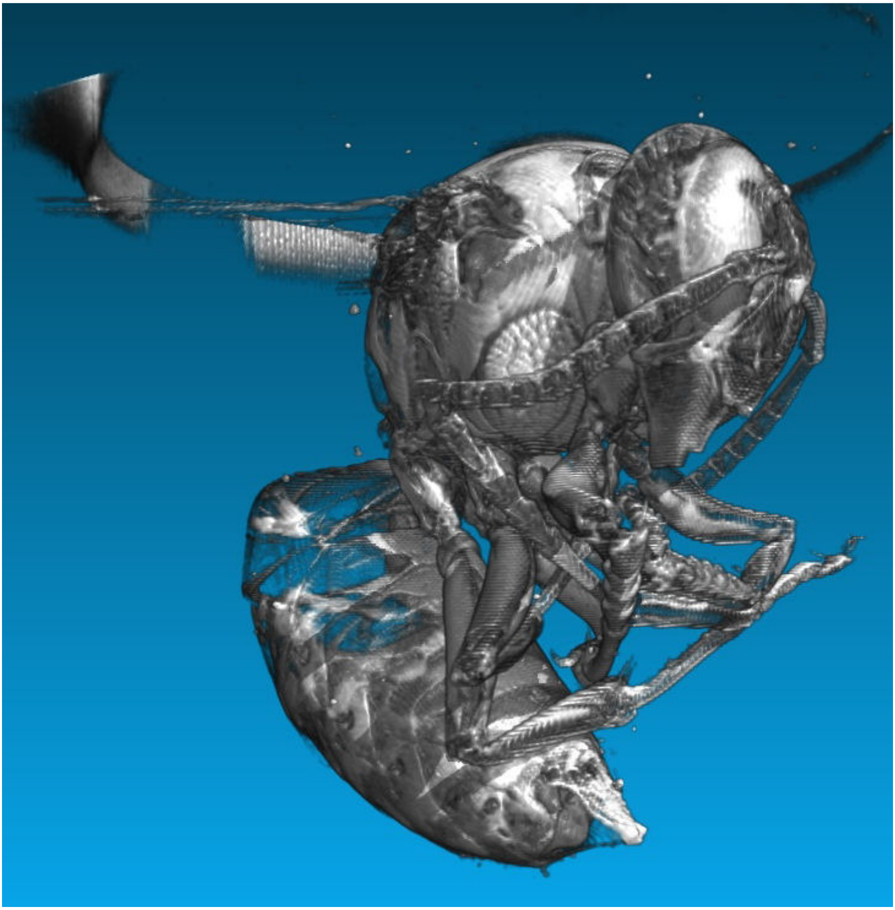
This function can also be activated via the button .



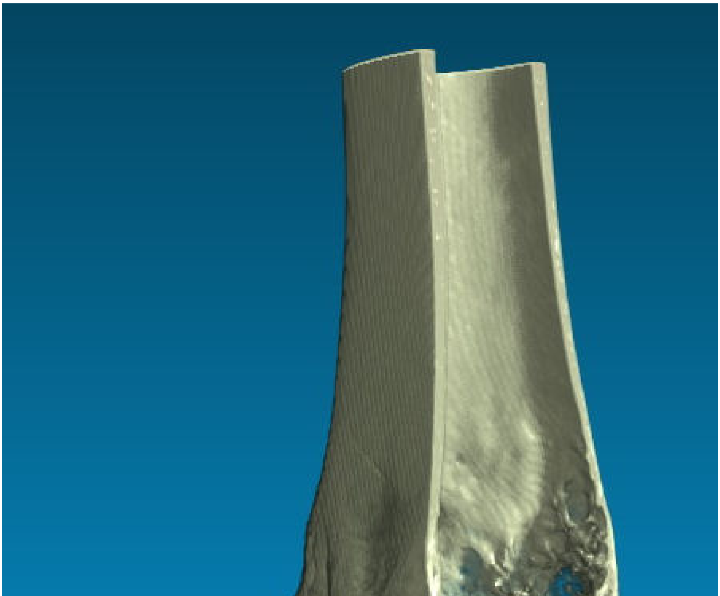
Example 1: Filament light bulb



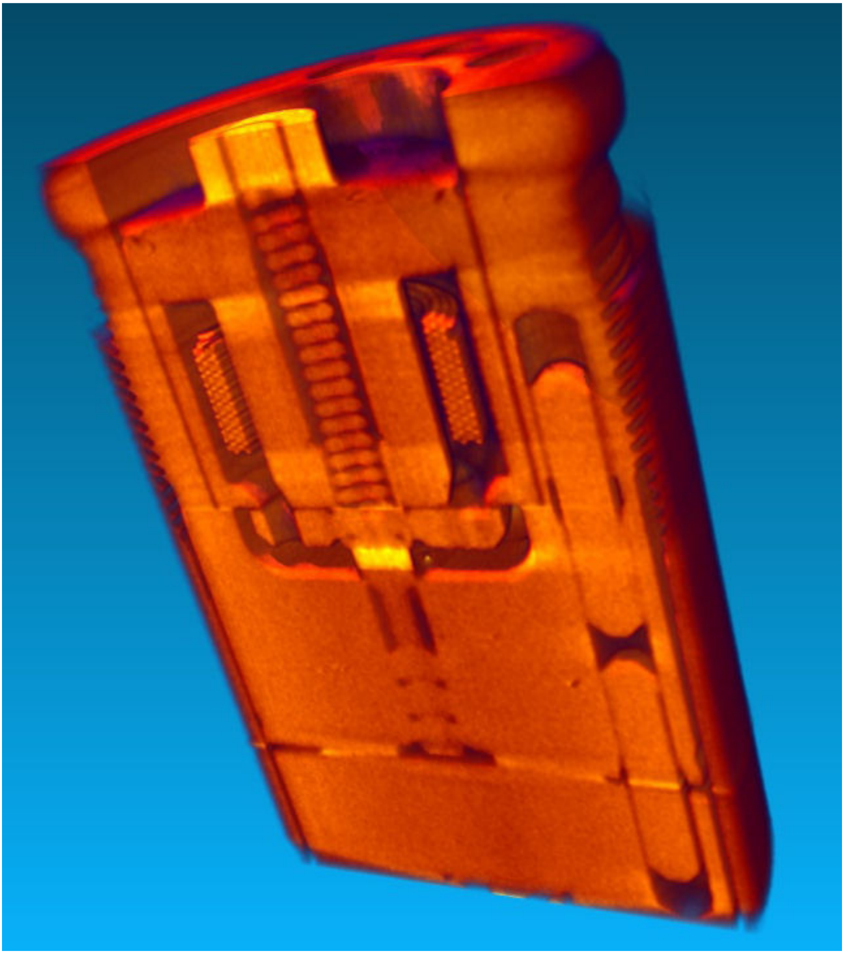
Example 2: Molar



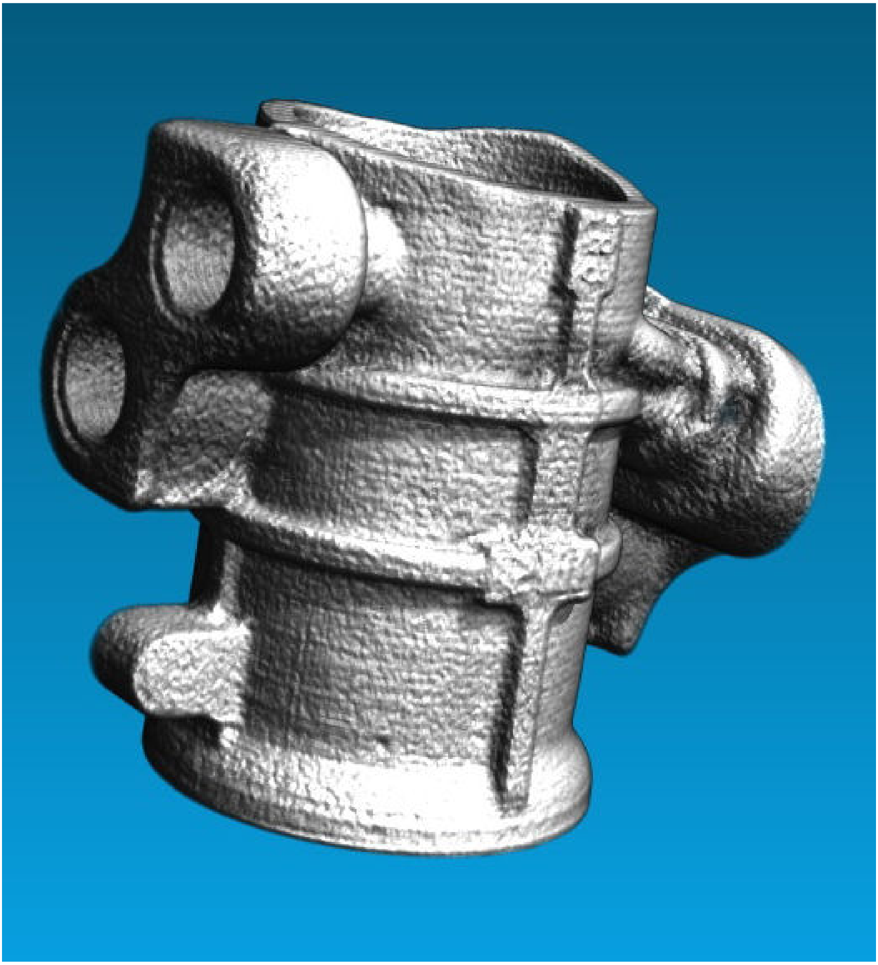
Example 3: Hornet



Example 4: Bone

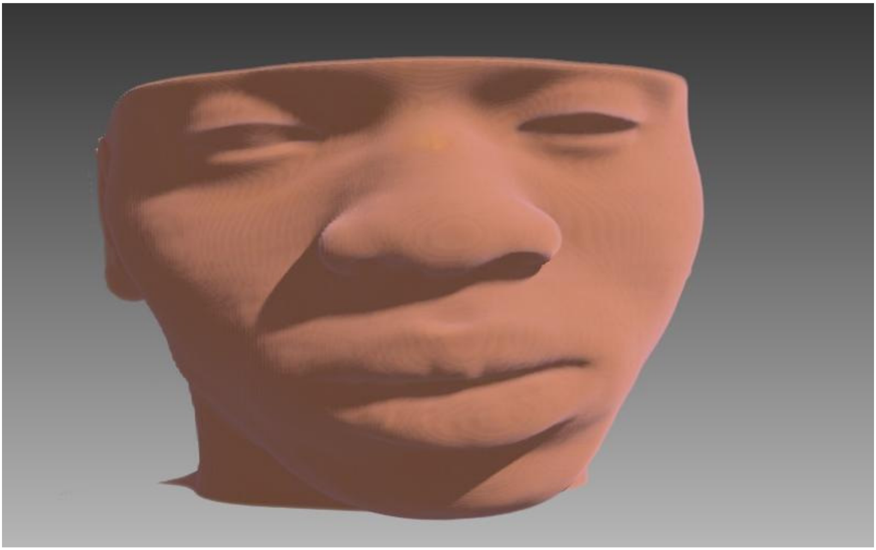


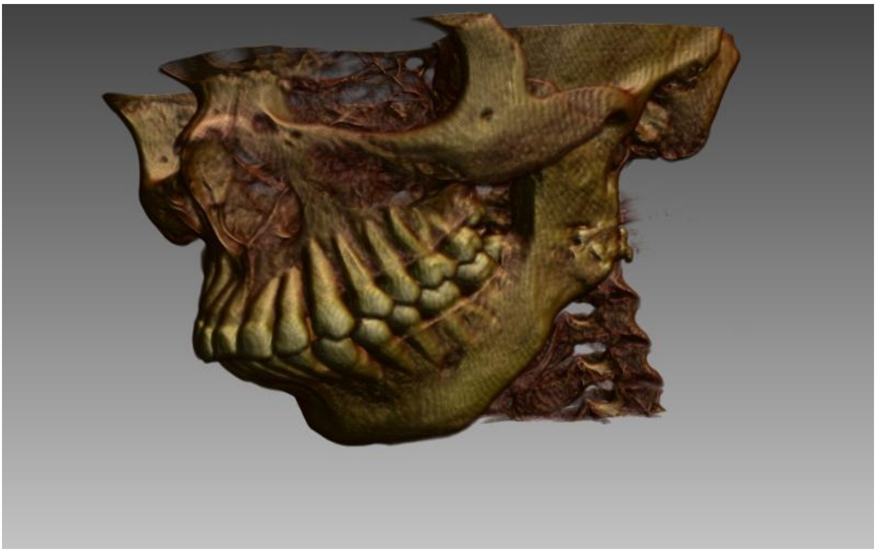
Example 5: Distribution pump



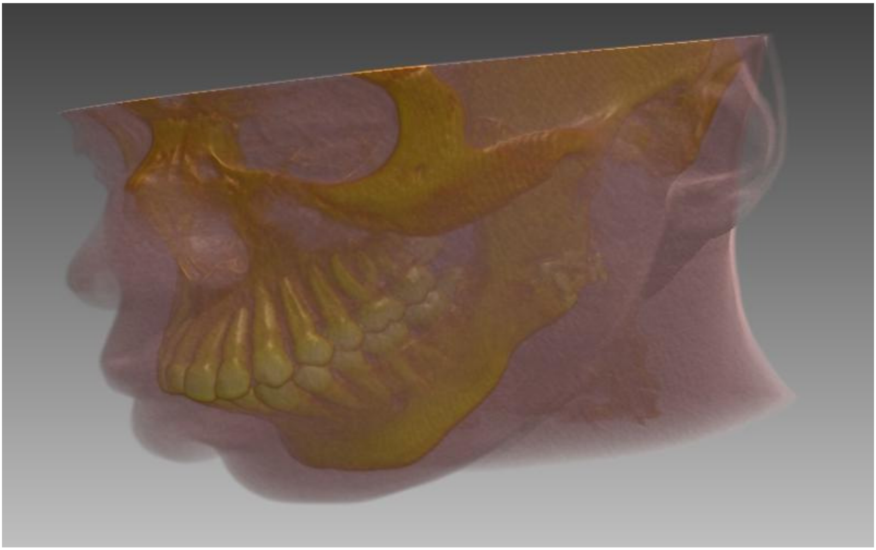
Example 6: Casing

Example 7: Talisman (museum)





Example 8: Human face and skull



Example 9: Combination of human face and its skull

3 Tomographic Reconstruction: DigiR3D Plug-in

3.1 Introduction

DigiR3D is the tomography reconstruction module of the DigiXCT software suite.

It produces the cartography of the attenuation of an object based on a stack of X-rays projections acquired from several angles of view.

DigiR3D commands are accessible in the `[Tools → 3D Reconstruction Parameters]` menu or the toolbar (Figure 3.1). This command opens the acquisition geometry configuration and reconstruction parameters dialogue box, and launches the reconstruction of the volume.

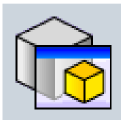


Figure 3.1: The `FDK Configuration` icon on the toolbar

3.2 Workflow definition

Figure 3.2 shows the DigiR3D GUI docked into the main DigiObs window. As can be seen, DigiR3D consists of 5 toolboxes (1: Hardware definition, 2: Input, 3: Output, 4: Reconstruction, 5: Preview/Adjustment). Those 5 steps are mandatory steps to perform a good reconstruction. This steps organization provides a clear vertical workflow to the user.

For each toolbox, you will find at least one tab corresponding to one mandatory group box and additional features enabled from expert mode or from license. Those tabs provide tools for specific application such as phase contrast tomography, or additional features to help reconstructing datasets (calibration tool). It also gives an access to image processing tools and iterative algorithms. For more details on features, please contact Digisens team.

Section 3.5 will describe a specific use of DigiR3D when the software handles X-ray manufacturer system geometry file (see system compatibility in this section).

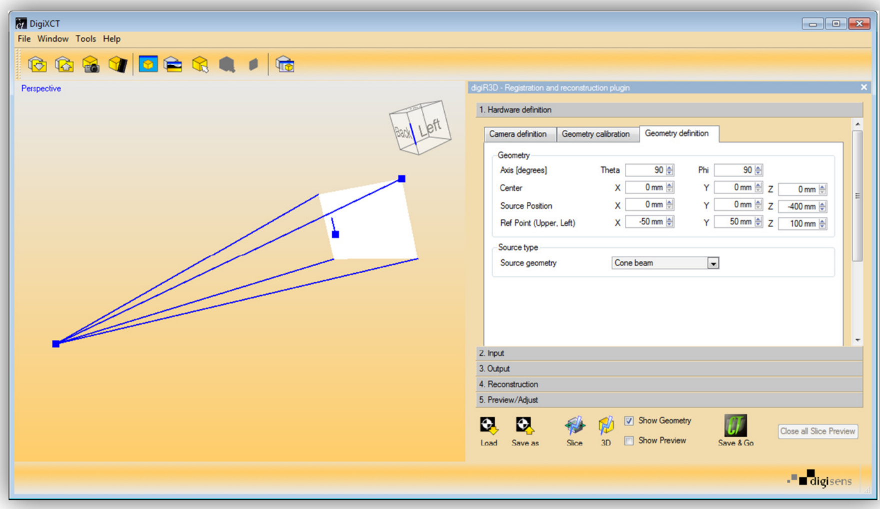


Figure 3.2: DigiR3D main window

3.3 Hardware definition

This toolbox aims to describe user's CT hardware.

3.3.1 [Mandatory] - Camera definition

In this first tab, you have to describe your imaging sensor. Several scenarios have been implemented:

- When DigiR3D is used for the first time, the GUI asks you to `create a camera` (see figure 3.3).
- When the camera has never been created, the software will automatically find the right camera definition and will fill the field in.

- When several cameras may be compatible with the currently loaded dataset, a popup will appear asking to choose the right camera.
- If the camera does not exist, the field is set to "`create camera`" and you will have to create it

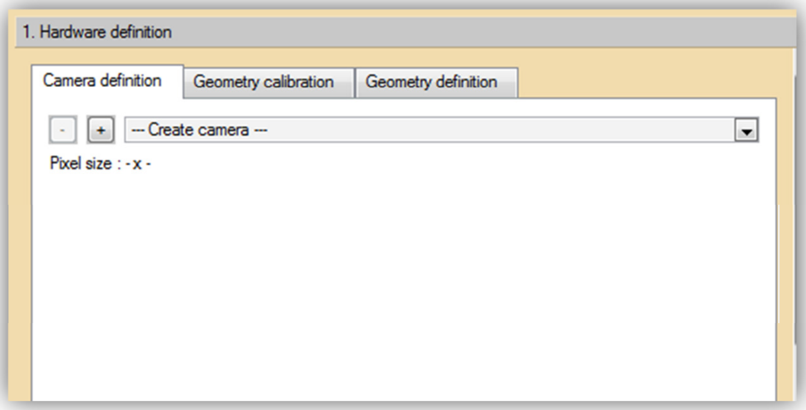


Figure 3.3: Camera definition tab

To manage a camera list, the `[+]` button allows a new sensor to be added, the `[-]` button to remove an imaging sensor. Figure 3.4 depicts the camera creation window. You have to give to your camera a name, its resolution in pixels and the pixel size. The third field will be automatically filled in.

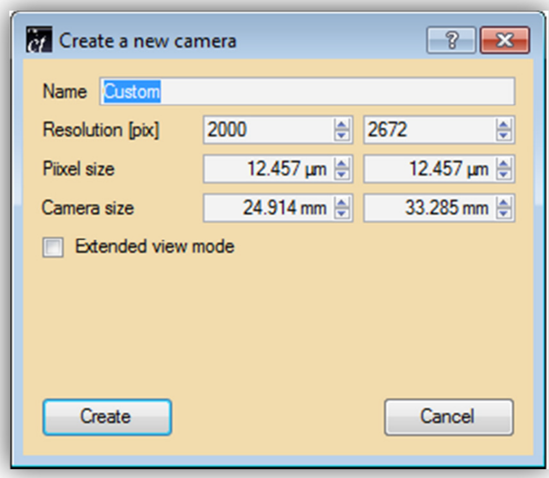


Figure 3.4: Camera creation

In some specific applications, in order to perform acquisitions of objects larger than the field of view, the system is equipped with a shifted detector as depicted figure 3.5. The mandatory condition to perform a good reconstruction with this acquisition setup is to keep an overlapping area between views of at least one third of the size of the detector width.

When the definition is correct, you have to click on `Create` button to return to the main interface.

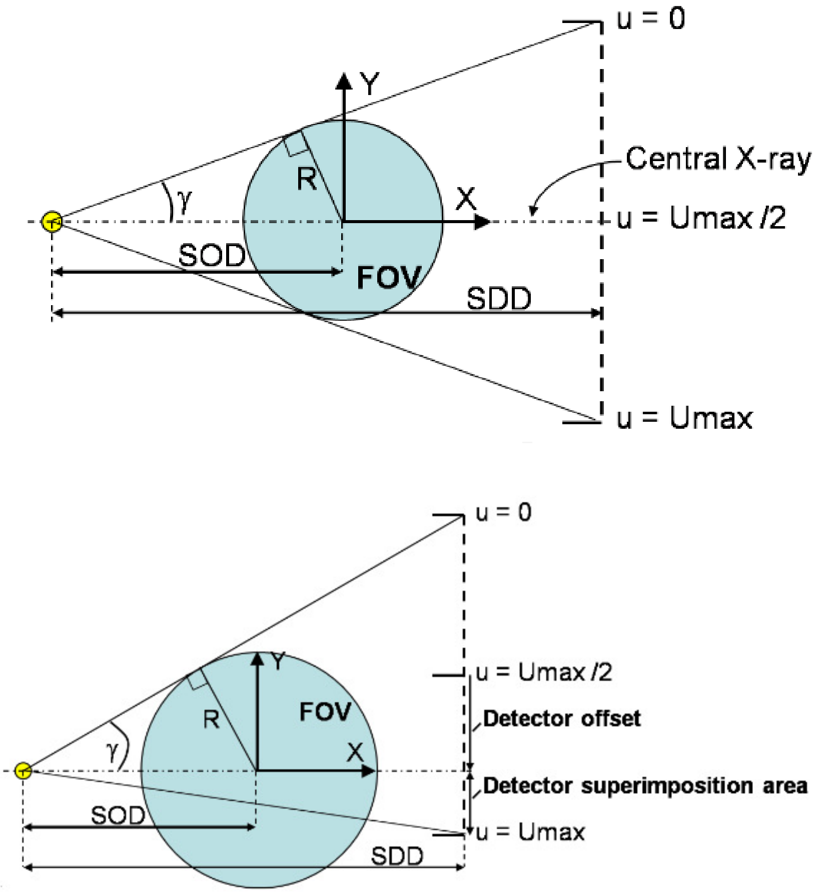


Figure 3.5: Extended view mode. Top illustration shows a standard geometry where the orthogonal ray from source to detector hits the detector center (for information, SOD corresponds to the source to object distance, SDD is the focal distance and Umax is the detector width). Bottom illustration shows the shifted detector.

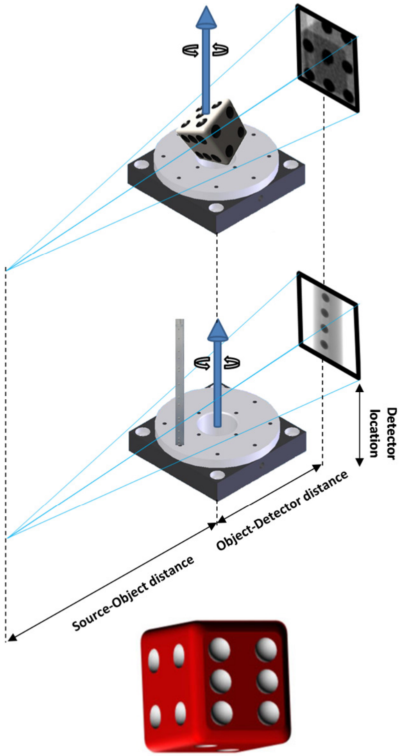
3.3.2 [Optional] – Geometry calibration

DigiXCT provides a calibration tool to determine required parameters to achieve a good tomographic reconstruction (rotation center, focal length, …) based on the capture of some radiographic acquisitions from a calibration test pattern. This test pattern is provided by Digisens. To get more information, please contact our sales team.

This feature is protected by the license.

3.3.2.1 Calibration procedure

A tomography scan including calibration step has to take place as follows:



1. X-ray acquisition of an object is carried out in the tomography system. The scan consists in the acquisition of a predefined number of images under different angles of view.

2. A test pattern, of size compatible with the previously scanned object, has to be acquired with the same system set up. A fast recording is carried out with a complete rotation, and a reduced number of images (a minimum of 60 projections, and always a multiple of 4).

3. The calibration plug-in estimates the geometry of the system based on the processing of previously acquired calibration dataset. At the end of the process, the user exports calibration results in a `*.cal` file.

4. Then, the software is able to

generate a 3D reconstruction of the scanned objet, ready for visualization and for further analysis.

In order to achieve good results, it is important to apply the following rules:

- ✓ Geometric settings of the CT system must not be changed between stages 1 and 2 (i.e. the position of the tray, the source and the detector must be maintained). Because of the mechanical tolerances of the robotics, the robotic arm never returns to the requested position, even if the accuracy of the positioning sensor is close to one micron.

- ✓ Additionally, if the object to be reconstructed is heavy, it is appropriate to put in a tare of approximately the same weight to compensate for the effects of robotic buckling. The buckling implies variations in the position of the rotation axis of several microns.

- ✓ Stages 1 and 2 can be executed in a different order. A whole series of objects can be scanned with the same calibration, but it is nonetheless prudent, in this case, to do at least calibrations, at the start and the end of the acquisition series so as to ensure that the machine has not shifted or been accidentally adjusted.

- ✓ At least 5 spots must be chosen, 10 are best. They must be at least one diameter's distance away from the edge of the image, and remain so during the whole revolution.

3.3.2.2 Calibration GUI

To reach the calibration interface, click on the tab `“Geometry calibration”` (See figure 3.6).

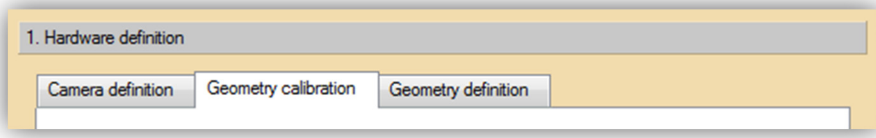


Figure 3.6: Calibration geometry tab

Steps for calibration:

- 1- Image loading (see figure 3.7):

To choose the series of images to be worked on, you only have to click on the first short cut of the toolbar: `Choose input images`. All images of a similar name (base name without image number) will then be selected.

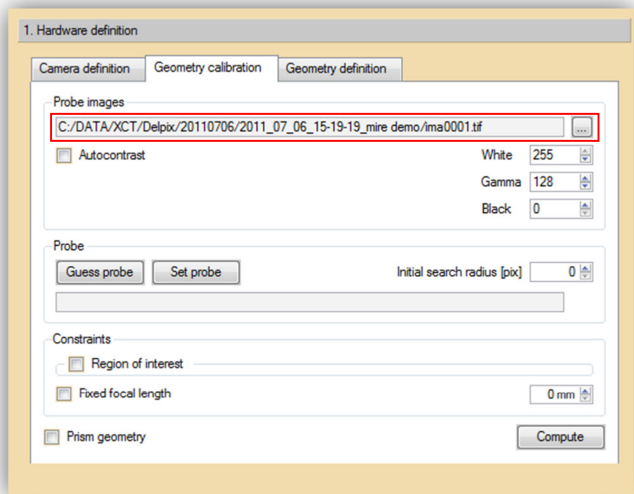


Figure 3.7: Image loading for geometry calibration

- 2 - Probe definition:

The `Probe` tool (see figure 3.8) allows defining the size of the probe, the relative position and number of spots.

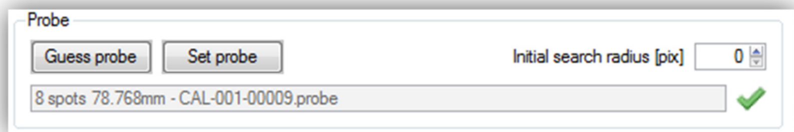


Figure 3.8: Probe definition.

First, it is necessary to indicate the dimensions of the spots in the `initial search size` field. A left click on the image of a spot launches the automatic search of its center and its radius. A click on an already selected spot, deselects it. And a double click deselects all the spots.

Note: The spots must be selected in order (preferable from the highest to the lowest on the screen) and never in a disordered manner.

Note: The selected spots must be visible in all the images and must never be partially hidden or cut off.

Important: In order for the trajectory of the probe spots to be monitored correctly, it is necessary that the initial speed (in the image plane) of the spots is low. (i.e. you start with the spots positioned completely on the left or the right of the image). You can reach this position via the cursor below the image. It is on this image that the initial position of the probe must be defined.

Finally, the probe can be validated by clicking the short cut `Set probe`, and by returning the distance between the center of the first and the last spot of the probe or by clicking on the button `Guess probe` if a calibrated probe is used.

When calibrated probes from Digisens are used and when beads are clicked, clicking on “`Guess Probe`” will automatically identify the probe and define the distance between the upper and lower spot.

The accuracy of probe size can be found in all distance measurements of reconstructed volumes: an overestimation of 1% (160 microns in our example) in the probe results in an overestimation of 1% on all size measurements of the reconstituted object.

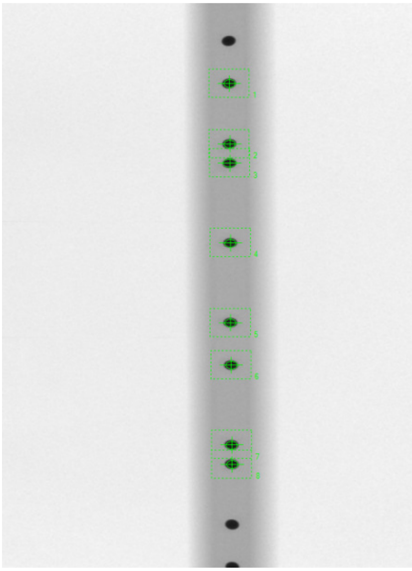
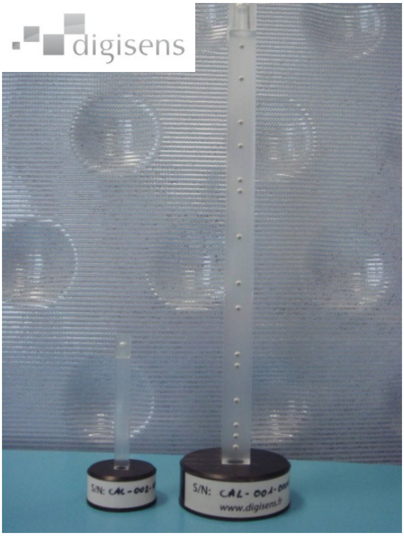


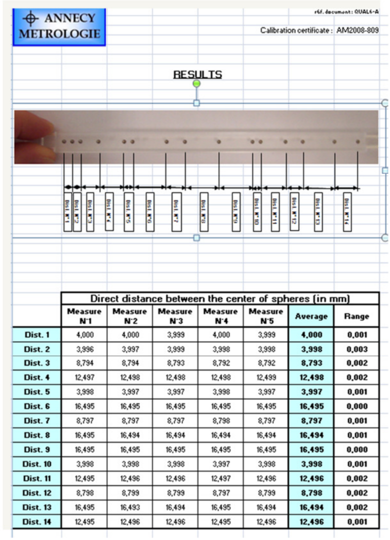
Figure 3.9: Spot selection

Digisens proposes to customers:

- Standard tools: a set of two probes. One probe is equipped with 0.4mm diameter beads spaced by 1.4mm. The second is equipped with 2mm diameter beads spaced by 5mm.

- As an option: calibrated probes with metrology inspection report (see figure 3.10). The probe and the report are unique. Those probes have two advantages: the tomography system is linked to metrology national references; they provide a simple tool for system calibration follow-up. The design is unique. Thus, calibration tool is able to automatically detect spots and provide the user with the right probe size using the button `Guess probe`. Using such tool provides a calibration repeatability lesser than 5µm, whatever system focal and user settings.




ANNECY METROLOGIE

réf. document : QUAL6-A

Calibration certificate : AM2008-809

RESULTS

	Direct distance between the center of spheres (in mm)						
	Measure N°1	Measure N°2	Measure N°3	Measure N°4	Measure N°5	Average	Range
Dist. 1	4,000	4,000	3,999	4,000	3,999	**4,000**	**0,001**
Dist. 2	3,996	3,997	3,999	3,998	3,998	**3,998**	**0,003**
Dist. 3	8,794	8,794	8,793	8,792	8,792	**8,793**	**0,002**
Dist. 4	12,497	12,498	12,498	12,498	12,499	**12,498**	**0,002**
Dist. 5	3,998	3,997	3,997	3,998	3,997	**3,997**	**0,001**
Dist. 6	16,495	16,495	16,495	16,495	16,495	**16,495**	**0,000**
Dist. 7	8,797	8,797	8,797	8,798	8,797	**8,797**	**0,001**
Dist. 8	16,495	16,494	16,494	16,494	16,494	**16,494**	**0,001**
Dist. 9	16,495	16,495	16,495	16,495	16,495	**16,495**	**0,000**
Dist. 10	3,998	3,998	3,998	3,997	3,998	**3,998**	**0,001**
Dist. 11	12,495	12,496	12,496	12,497	12,496	**12,496**	**0,002**
Dist. 12	8,798	8,799	8,799	8,797	8,799	**8,798**	**0,002**
Dist. 13	16,495	16,493	16,494	16,495	16,494	**16,494**	**0,002**
Dist. 14	12,495	12,496	12,496	12,495	12,496	**12,496**	**0,001**

Figure 3.10: Calibrated probes.

- <u>3 – Optimizations (optional):</u>

The `Region of interest` section (see red and green borders figure 3.12) allows the image to be cropped, which is rarely useful, other than when some spots get too close to an image border leading to a tracking failure. The `uMin, uMax` values therefore represent the horizontal interval used, and `vMin vMax` the vertical interval.

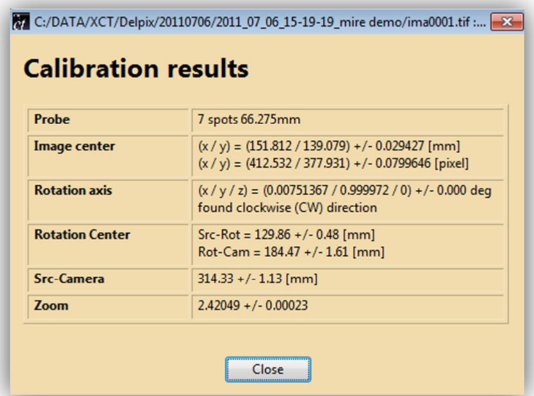
It is also possible to fix focal distance value as a prior knowledge to improve geometry estimation accuracy.

- <u>4 – Geometry estimation:</u>

Once the probe has been specified, the `Compute` button allows the calibration calculation to be launched. This calculation can fail (red marking on one or several spots) for the following reasons:

- ✓ The initial speed of the spots is too high: the processing must be started on an image where the spots are completely on the left or on the right of the field of view.
- ✓ The speed is not regular: an image must have jumped or been processed twice. The calibration sensing procedure must be re-visited in such a way as to have regularly spaced out images.
- ✓ A spot collides with the edge, and the automatic search detects a sharp variation in spot diameter. The parameters `uMin, uMax, vMin` or `vMax` must be used to trim the images, or this spot should not be selected for the calculation.
- ✓ In a particular angle (or in a certain part of the sensor), the probe support is too contrasted which prevents the spots from being properly detected. The contrast setting must be re-visited.
- ✓ The most horizontal trajectories tend to introduce greater centering errors. The (one or two) spot(s) nearest the center of the image should be deselected.
- ✓ A spot can shift in the probe during sensing. In this case, the trajectory will be badly centered. It should not be selected in the probe.

When the calculation is successfully completed, calibration tool posts a results table (see figure 3.11) and spot trajectory is displayed (see figure 3.12).


C:/DATA/XCT/Delpix/20110706/2011_07_06_15-19-19_mire demo/ima0001.tif :...

Calibration results

Probe	7 spots 66.275mm
Image center	(x / y) = (151.812 / 139.079) +/- 0.029427 [mm] (x / y) = (412.532 / 377.931) +/- 0.0799646 [pixel]
Rotation axis	(x / y / z) = (0.00751367 / 0.999972 / 0) +/- 0.000 deg found clockwise (CW) direction
Rotation Center	Src-Rot = 129.86 +/- 0.48 [mm] Rot-Cam = 184.47 +/- 1.61 [mm]
Src-Camera	314.33 +/- 1.13 [mm]
Zoom	2.42049 +/- 0.00023

Close

Figure 3.11: Calibration result example

The program cannot find a solution with less than 3 spots. In practice, at least 5 or 6 are necessary to have error calculation results. Ten or so covering the whole of the area to be reconstructed would seem to be optimal.

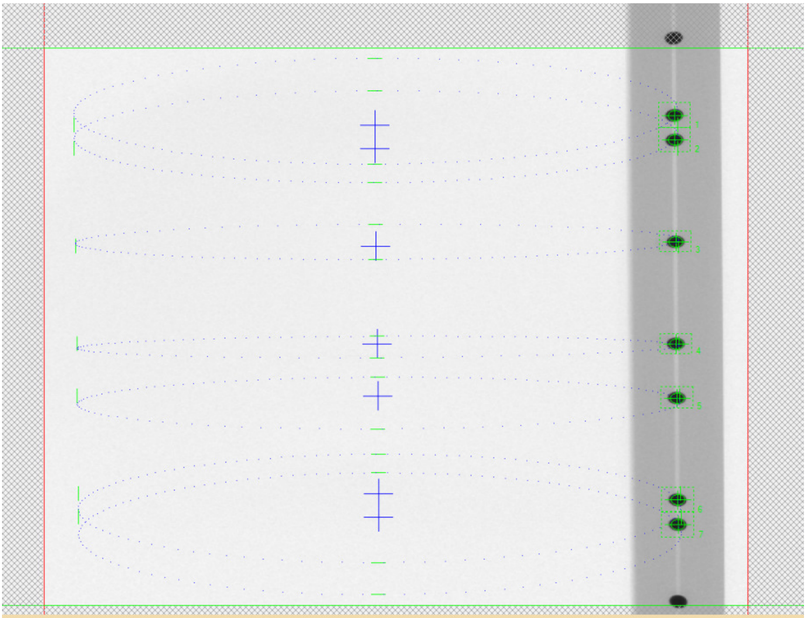


Figure 3.12: Spots trajectory

3.3.3 [Mandatory] - Geometry definition

The tab `Geometry` describes the geometry of the system used to acquire the dataset.

The following figures depict the definition of geometry parameters.

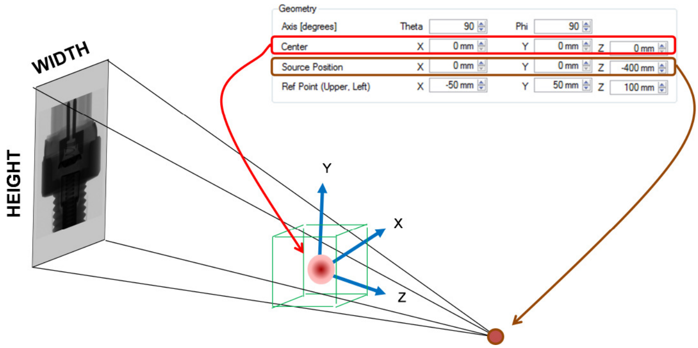


Figure 3.13: Source, axis and geometry center definition

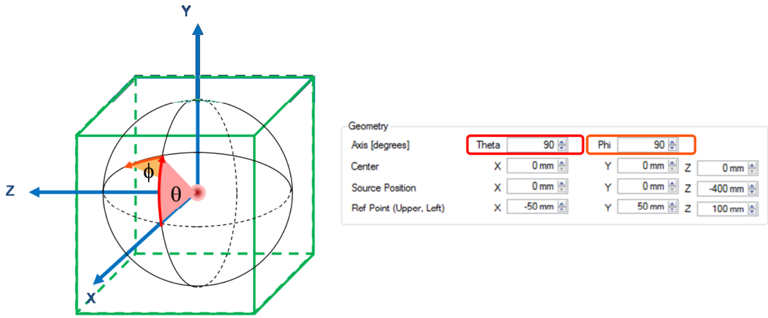


Figure 3.14: Spherical coordinate system definition

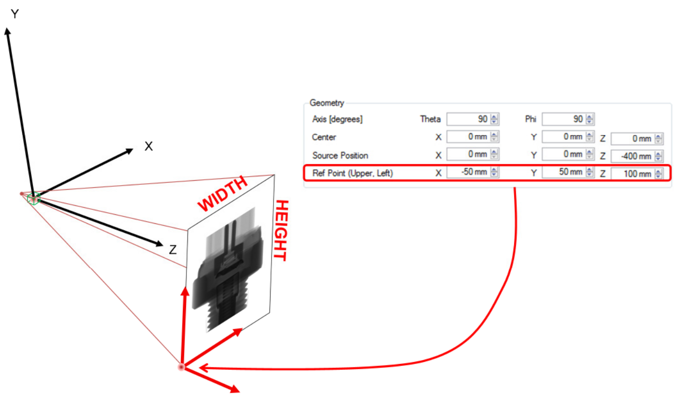


Figure 3.15: Reference point definition

DigiR3D handles several beam profiles. You can select them from the combo box figure 3.16.

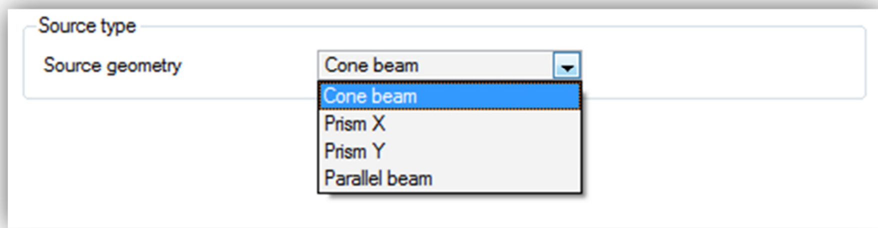


Figure 3.16: Beam profile selection

Specific case of fan beam reconstruction:

- Select a dataset in multipage raw format, this means a single file containing the sinogram in the following arrangement widthx1xheight
- Define raw file in DigiR3D GUI
- Select PrismY geometry (this means divergent beam horizontally and parallel beam vertically.

3.4 Input

This second toolbox aims to describe and to process projections used to achieve a correct reconstruction.

3.4.1 [Mandatory] - Input definition

The `Input files` field (see figure 3.18) indicates the directory where the projections are located (that can also be selected using the explorer by clicking on the button ).

The image formats supported are: TIF, DICOM, VFF, ATN, RAW, JPG, BMP….

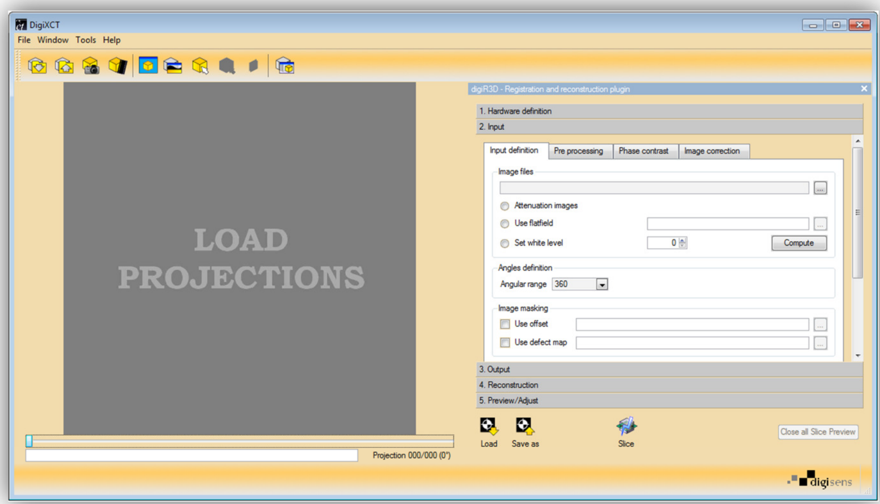


Figure 3.17: Input tab presentation

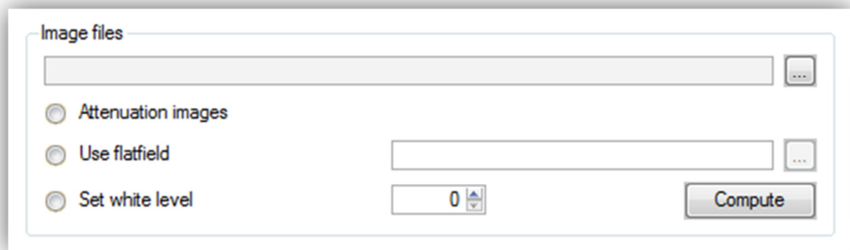


Figure 3.18: Input files

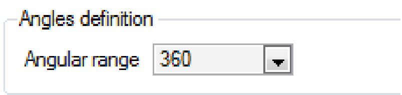
According to acquired image type, several scenarios have been implemented:

- If images are intensity-based images, you have to use a flat field image (also known as bright image, beam profile image …). To

use it and to inverse the Beer Lambert law, just select it using the explorer by clicking on the button). Automatically, the software will compute an attenuation image by dividing the flat field image by the acquired projection and by computing the logarithm of the resulting image.

- If the flat field image does not exist, you can generate a constant image with a level corresponding to the mean value of the bright image. This level can be measured on a small region of interest where the object is not present or you can use the mean value of the entire image stack. If this measure is impossible, just let a 0 value in the field set white level.
- For some systems, the acquisition software generates automatically attenuation images (meaning that the correction from flat field image, the offset image and the computation of the log is already done). If this is your case, just click on `Attenuation images` radio button.

Once the images have been loaded, you must indicate to the software the angular range. This can be done by filling the field `Angles definition` in. DigiR3D can handle full scan (angular range of 360 degrees) or partial scans acquisitions (at least 180 degrees + beam divergence). This field can be edited meaning that you can write any value in it.



You have the opportunity to make some additional correction on images. When the check box `Expert user` is checked, you can compensate images from electron noise offset by selecting the corresponding image (see figure 3.19).

`Use defect map` correction consists in eliminating pixels or pixel clusters with abnormal response (it means with null or saturating response whatever signal received). In tomography field of application, this correction is mandatory to remove the well-known ring artifact. The image used for this processing has to be binary: white level for abnormal pixels and black level elsewhere.

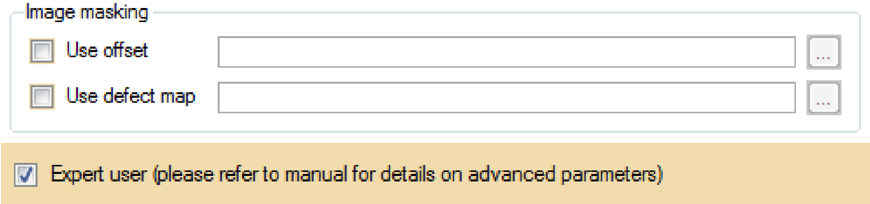


Figure 3.19: Additional image correction

Important note: Projections displayed are as they will be used in reconstruction step. This means that projections background has to be dark and the part brighter.

3.4.2 [Optional] – Projection pre-processing

Several filters are available to pre-process projections before using them in the reconstruction process.

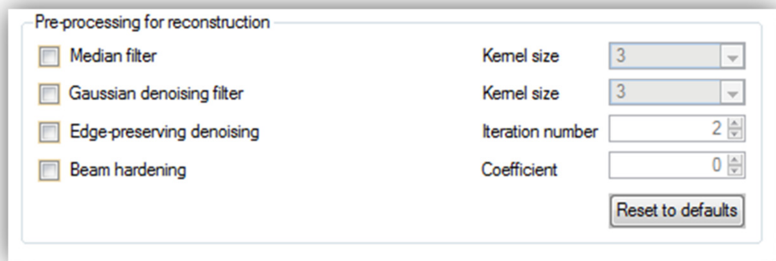


Figure 3.20: Image preprocessing list

3.4.2.1 Median filter

Median filter is applied on projections. It is often used for noise reduction on images. This filter is well known for its performances as "salt and pepper" noise remover. In tomography, it will be very useful to eliminate the dead pixels (always saturated or always switched off) or pixels with a singular behavior. Their effect will be the introduction of ring artifacts.

The median filtering consists in replacing a pixel by the median value of a sorted list of the nearby pixels value. The sorting is made from the smaller to the higher value. The number of pixels of its neighborhood is defined by default in a value of 1 pixel on both sides of the pixel to be corrected (3x3 kernel). The size of the kernel can be increased to 5x5 maximum.

The choice of kernel size requires a particular attention on behalf of the user. As this filter replaces the current value by the value of one of the neighboring pixel, it introduces a distortion in the image. As a consequence, a filter with a too large kernel size will tend to degrade strongly the resulting image.

The figure 3.21 illustrates this.

3.4.2.2 Gaussian filter

Gaussian filter is applied on projections. It is often used to reduce noise in images. The principle of this filter is to average the pixel with its neighborhood. The weighting applied to every neighbor takes into account the distance of the neighboring pixel to the processed pixel. This filter removes the noise, but ads blur to the corrected image.

The number of pixels constituting the neighborhood is defined by default in a value of 1 pixels on both sides of the pixel to be corrected (filter kernel of size 3x3).

In expert mode, the kernel size can be adjusted.

The figure 3.22 compares the various techniques of filtering.

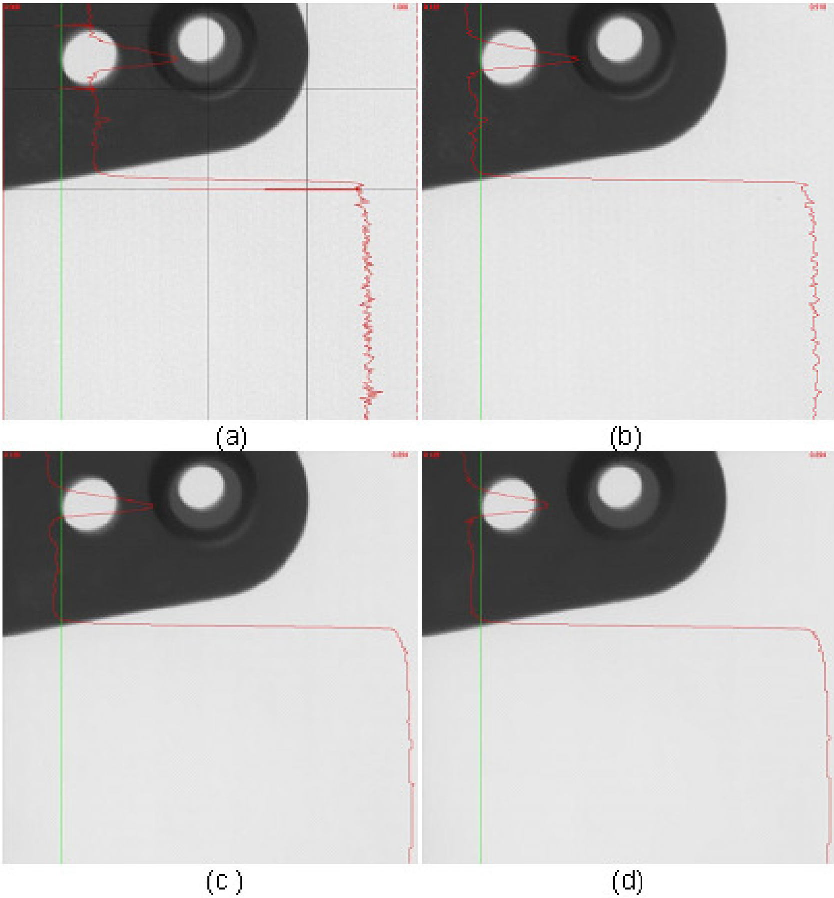


Figure 3.21: Median filtering with variable kernel size on image containing interesting details (a). Filtering with a kernel size of 5 pixels (b), 20 pixels (c) and 40 pixels (d). Intensity profile plotted shows modulations in the part corresponding to an interesting defect in the part. It is obvious that median filter with too large kernel size degrade information.

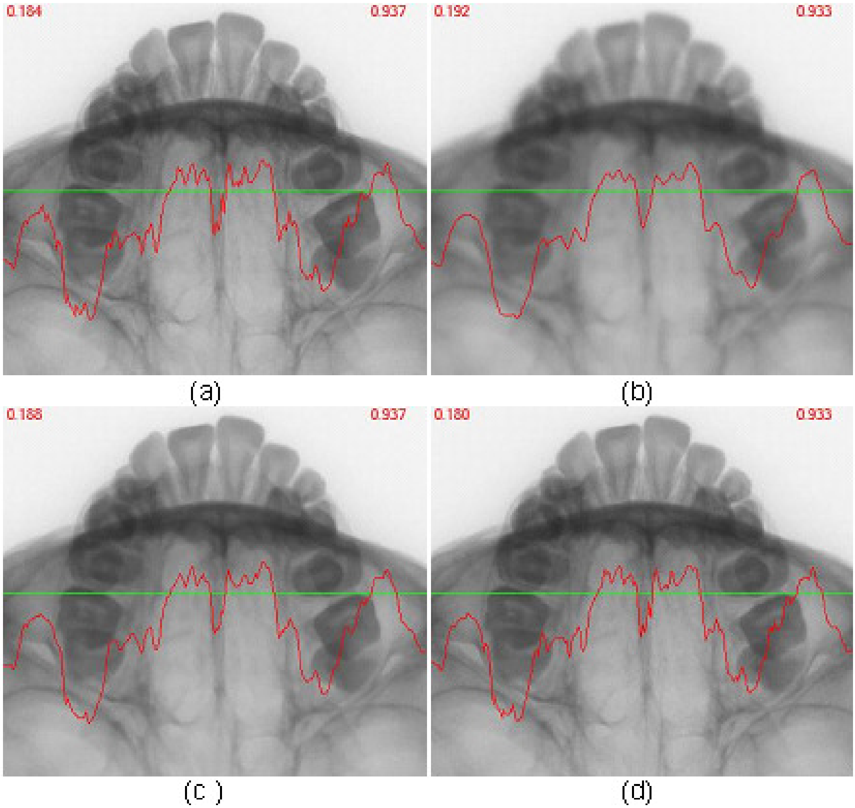


Figure 3.2: The image (a) is filtered by a gaussian filter of size 5x5 (b), by a median filter of size 5x5 (c) and by an edge preserving with 2 iterations (d). We can see that the median protects details compared with the gaussian filter, and that the edge preserving filter offers performances on top of these two filters, while allying the interest of both filters.

3.4.2.3 Edge preserving filtering

To avoid Gaussian filter limitations (blur), a filter preserving discontinuities is implemented. This filter is recommended if projections have to be filtered keeping a maximum details on object transitions. This filter is iterative. The only parameter to adjust is the number of iterations, in

expert mode. The more iterations, the smoother the images. By default the parameter is 2 iterations.

The figure 3.22 compares the various techniques of filtering.

3.4.2.4 Beam hardening compensation

The beam hardening coefficient (`Beam hardening`) allows the artifacts due to the polychromacity of the source (several lengths of wave) to be corrected): depending on the thickness of the materials passed through by the x-rays and the source energy, the artifacts appear in the form of an abnormally light grey in certain zones, especially around the edges as illustrated in Figure 3.23. The values to use are from 0 (no compensation) to 10 (strong beam hardening). For higher values, the image may disappear.

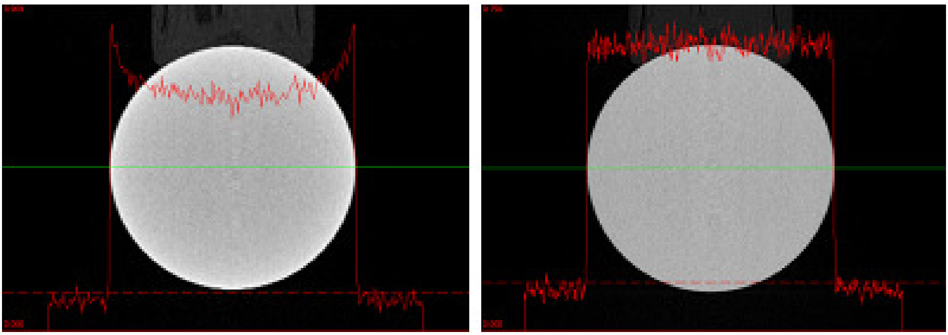


Figure 3.23*: Beam hardening artifacts before and after compensation*

3.4.3 [Optional] – Phase contrast pre-processing

Phase contrast pre-processing allows the reconstruction of a set of mixed projections. This kind of projection contains both absorption and phase contrast. A mixed projection can be easily recognized when object components edges are brighter than regions surrounding. Without any processing, this phase component introduces severe artifacts in reconstruction when restoring object edges (See figure 3.26). The implemented approach is based on Bronnikov based correction.

This module is under license.

Image correction has to be performed in two steps:

- First step, apply Bronnikov filter on image to split pure absorption signal from phase signal. Choose a value that leads to a low-frequency representation of the original image (See figure 3.24, 3.25)
- Second step, fix the BAC correction such as you are close to contrast inversion and with a value lower than MBA correction parameter. Compared to BAC image, this one does not show any phase effect and high frequency information is included (See figure 3.24, 3.25)

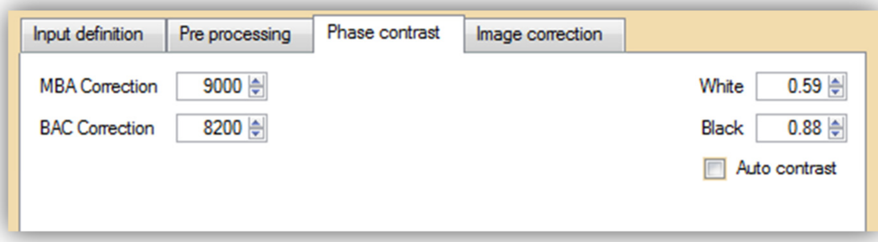


Figure 3.24: Phase contrast filtering parameters

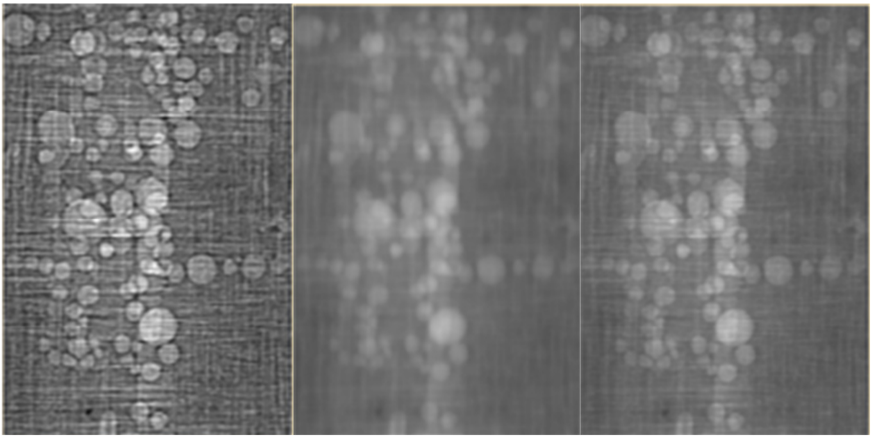


Figure 3.25: Effects of MBA and BAC corrections on the projection of a carbon fiber plate.

Figure 3.26 shows the corresponding reconstruction, with and without phase compensation.

3.4.4 [Optional] – Projection correction

During a tomography scan, the flux emitted by the X-ray tube varies (generator variations, generator or tube regulations ...). The consequence is contrast variations in acquired images. As long as it is possible to monitor these variations, it is possible to correct them. **This processing requires an area in the image in which the object will never be acquired; otherwise the correction is unfortunately impossible with this tool.**

To process image sequence, check the box `Flux Variation` (see figure 3.27). This processing need a reference image used as a flux reference for all other images in the sequence. Ideally, this image should be a white image (image acquired without object and with the same conditions). To display this image, check the box `display reference image`.

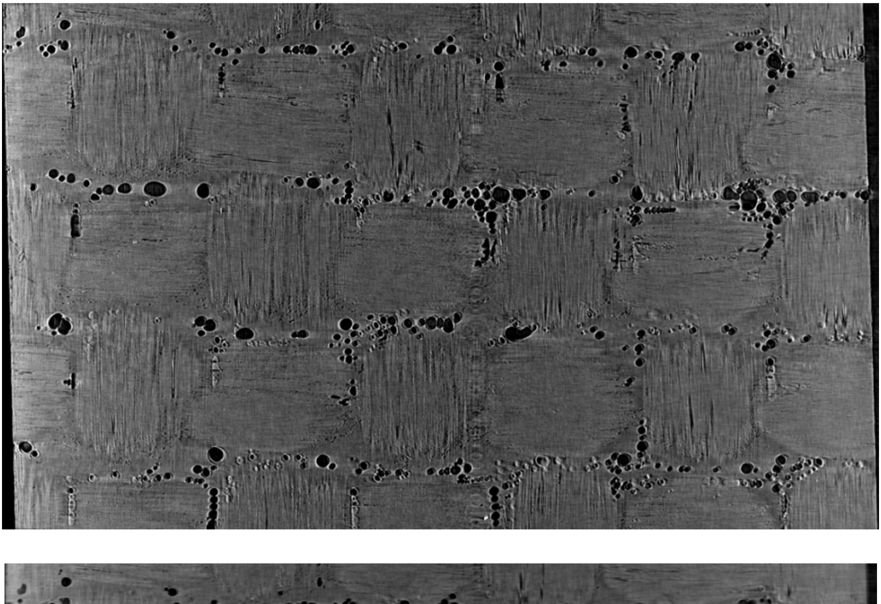


Figure 3.26: Top image corresponds to a reconstructed slice of a carbon plate with phase effects. Bottom image corresponds to the same reconstructed slice with BAC approach.

Then, define the ROI for measurement by modifying the red square extensions. Just modify its width and height, and locate it in the region you need to monitor

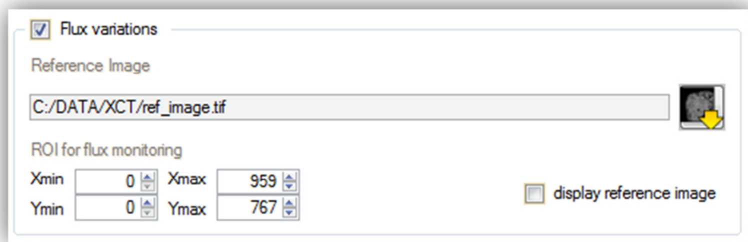


Figure 3.27: Flux variation correction parameters

After X-ray emission begins, the target of the tube is warming. Because of this temperature increase, the X-ray spot drifts leading to projection shifts. The consequence on high resolution reconstruction is straightforward and significant for scans longer than 1h. According to scan duration, this drift is assumed to be linear. In this case, it is easy to compensate its effect using an extra image acquired at 360° and using a linear compensation model. For very long scans, drift is more complex and need more than one image.

To use this drift compensation, load the extra image in the dedicated field and click on `compute` button to estimate de drift magnitude. Then all projections will be shifted to compensate from this drift. If one wants to cancel the compensation, just click on `clear` button.

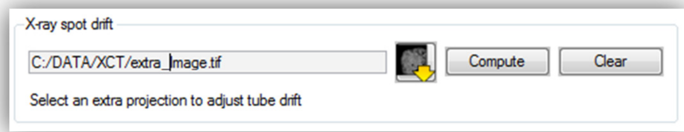


Figure 3.28: X-ray spot drift correction parameters

Note: the software is compatible with Skyscan drift compensation. To use all those extra images, just load the first iif file from Skyscan dataset.

3.5 X-ray CT system compatibility

DigiR3D is compatible with the following CT system manufacturers:

- GE Measurement& control solutions - Phoenix:
- GE Healthcare for small animals imaging
- Skyscan
- Nikon Metrology – XTek
- RX Solution
- Cyx+
- Yxlon
- NSI

For a detailed list of compatibility, please contact Digisens commercial team.

If the CT is handled by DigiXCT, when the user clicks on load, the file browser opens and the user will be able to choose the geometry file associated to the CT system as depicted in figure 3.29.

If the system is not handled by the software, the user can fill all necessary fields based on the procedure described from chapter 3.3 to chapter 3.4.

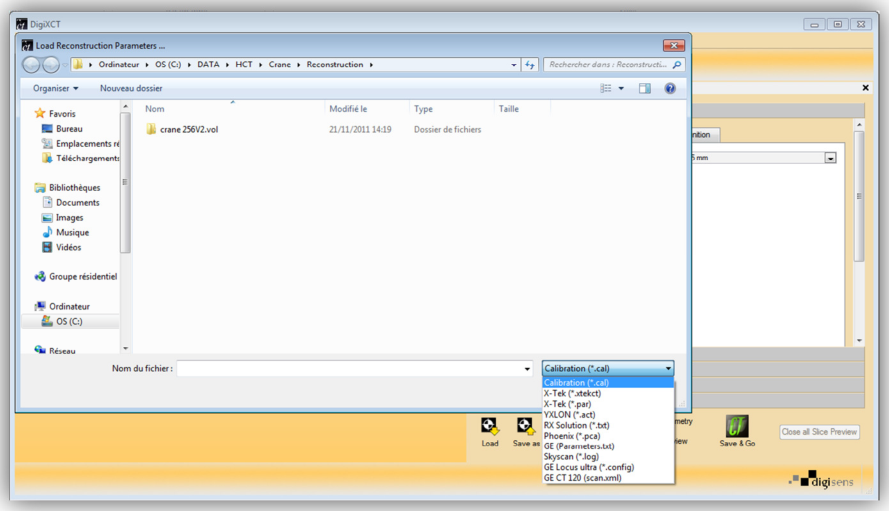


Figure 3.29: Compatible CT system list

3.6 [Mandatory] - Output definition

3.6.1 Volume format and name

Two formats are handled by DigiR3D:

- Image stack format (format v0 with extensions tif, raw, dicom, jpg, bmp…)
- Digisens' 3D file format (volume v2)

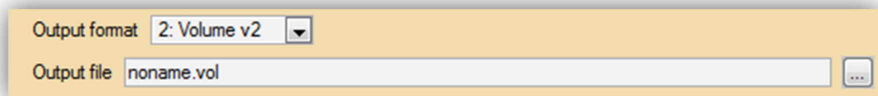


Figure 3.30: Projections registration

To define the name and the location of saving of reconstruction file, you have to click on the button to open a file browser and write necessary information.

3.6.2 Volume dimensions to be reconstructed

The `Dimension` field allows the name of the file containing the reconstruction data and the directory in which they will be stored to be specified. The volume dimensions (in mm) are defined by the center of the 3D grid to reconstruct (`Center`) and the size of the grid (`Size`) shown in red (see figure 3.31). These two values can be automatically positioned (`auto` box) in order to reconstruct an optimal volume in relation to the data supplied, or they can even be adjusted manually in order to only reconstruct a chosen zone.

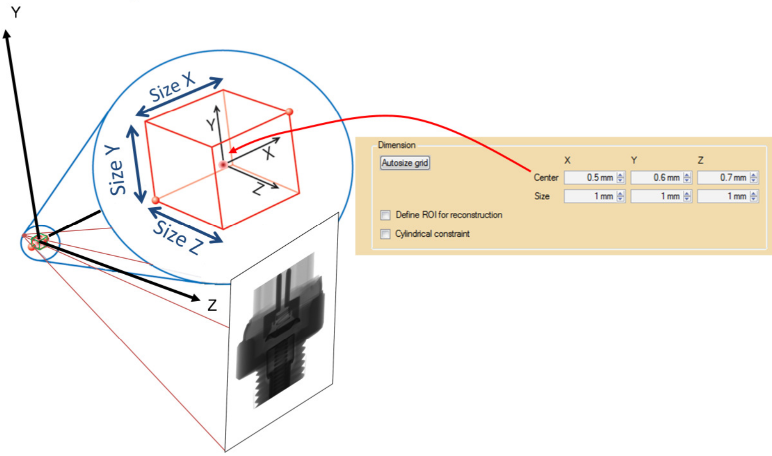
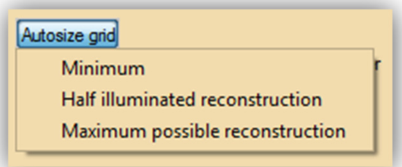


Figure 3.31: 3D reconstructed volume definition

The button `AutosizeGrid` displays a menu with 3 options described below:



`Minimum:` reconstruction algorithm will take into account only regions in the volume visible in all projections.

`Half illuminated:` reconstruction algorithm will take into account regions in the volume that are visible in at least half of the dataset.

`Maximum possible reconstruction:` the entire volume will be reconstructed.

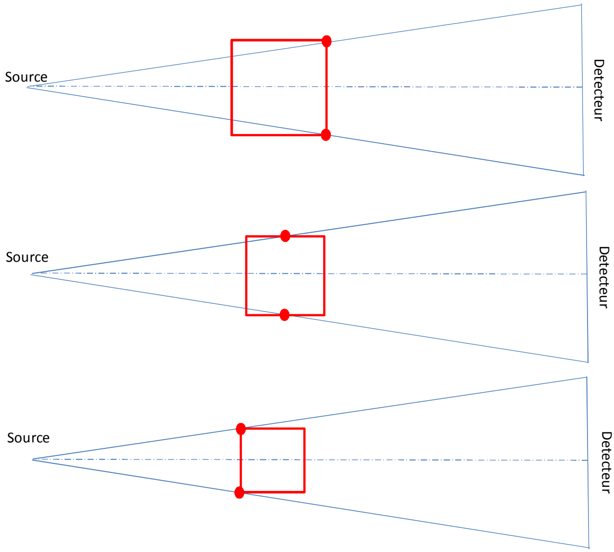


Figure 3.31: Autosize volume definition. From the top: *`Maximum possible reconstruction, Half Illuminated Reconstruction, Minimum`*

Note: when export mode is invalidated, `Autosize grid` button fixes dimensions to `Half Illuminated Reconstruction`.

Note: intuitively, we could expect that default dimensions proposed by the software have in width and height the resolution of the detector. The figure 3.33 illustrates the principle of estimation of the dimensions. In a conical context, the reconstruction will be made in the cylinder registered in the cone of the beam defined by the dimensions of the detector. The reconstruction grid contains this cylinder. The figure shows that in a general conical case, the cube is narrower here than what it would be if the beam divergence was weaker because the inscribed circle has a diameter smaller than in a case where the beam divergence is weak.

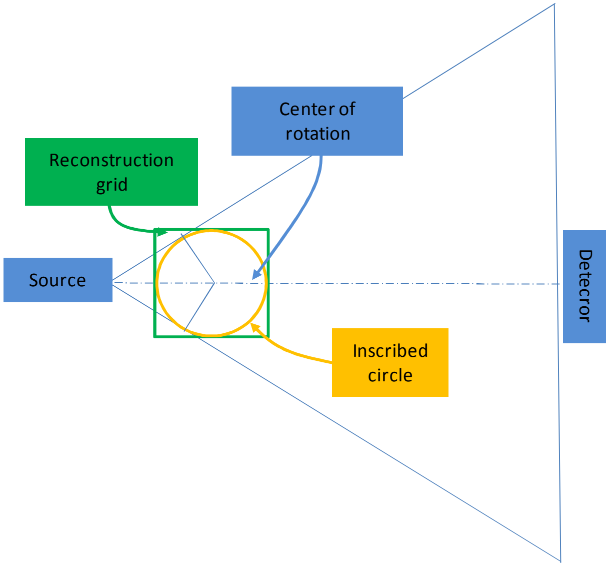


Figure 3.33: Determination of reconstruction grid dimensions

3.6.3 Region of interest

You can reconstruct a region of interest in your volume (see figure 3.34). To do this, you can either return the dimensions of the area of interest using the DigiR3D interface, or by using the multi-display visualization described in chapter 2.4 and defining the area of interest by using the mouse. In this latter case, the `Define ROI for reconstruction` box must be ticked.

The dimensions of the region can simply be changed by left-clicking on the boundaries, and whilst keeping the button pressed down, gently dragging to change the boundaries. The 3D visualization window which defines the volume is thus updated. To refine the definition of the area of interest, it is possible to zoom in by pressing the keyboard's `ctrl` button and by using the mouse's roller wheel. Returning to the original zoom is achieved by pressing on the keyboard's `ctrl` button and by left double-clicking on the mouse.

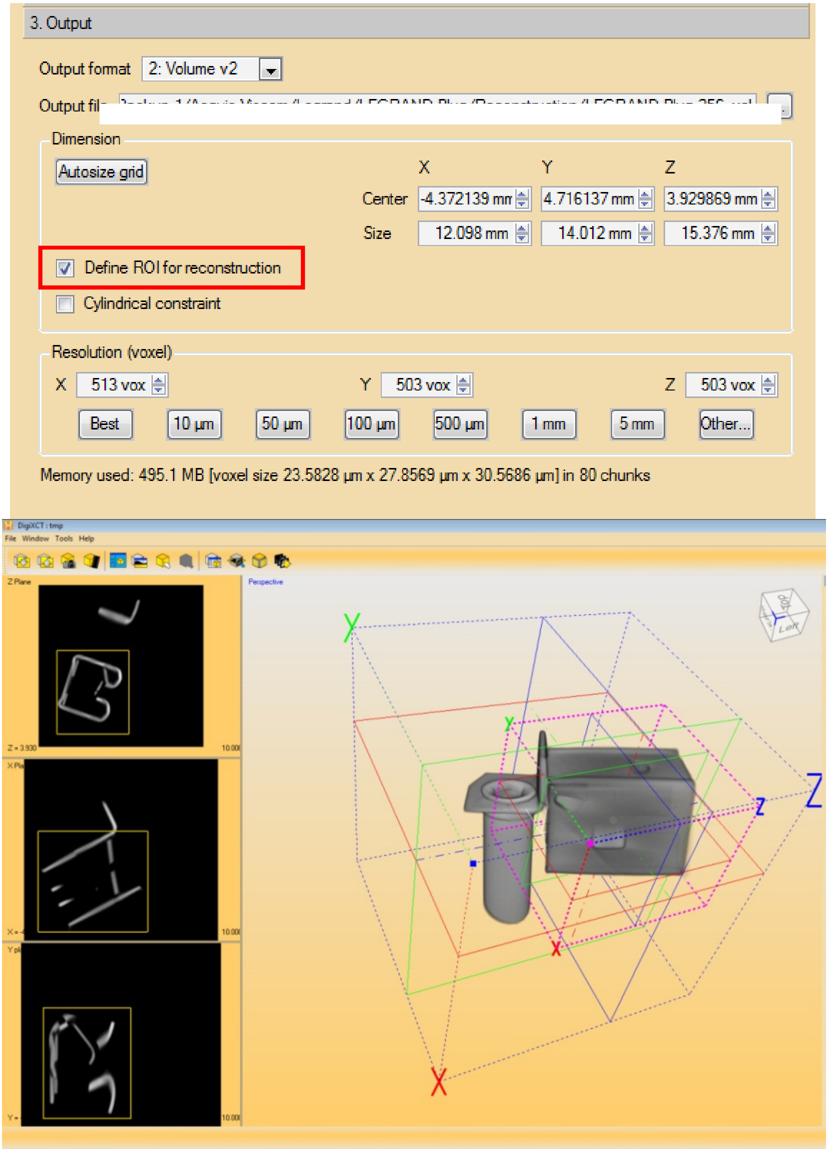


Figure 3.34*: Definition of the reconstruction area of interest*

3.6.4 Volume resolution

The resolution of the volume for each of its dimensions is shown figure 3.35.

You can also directly set the resolution of the reconstruction either by clicking on the buttons corresponding to a pre-defined resolution value, or by choosing `Other` which allows a value to be put in whatever the resolution. Once the resolution has been chosen, object dimensions in voxels are updated.

The resolution of the volume on each of the dimensions is indicated (see figure 3.35). So reconstructed voxels are not necessarily of the same size on the three axis.

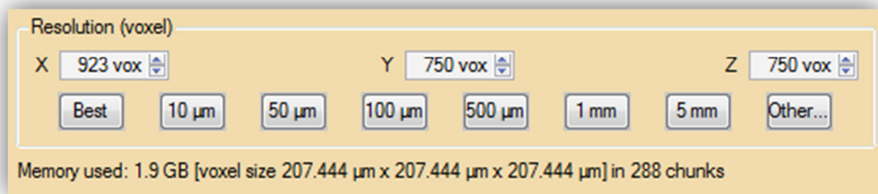


Figure 3.35: Volume resolution

3.6.5 Cylindrical reconstruction constraint

The cylindrical constraint allows limiting the reconstruction to a region of the space corresponding to the intersection of the back-projection of all the available projections. This intersection is centered on the axis of rotation. Outside, data corresponds only to the intersection of some beam rays common to some projections. They are thus carriers of information with weaker signal to noise ratio. The information is thus degraded by a stronger noise and the levels of density can be biased. The figure 3.36 illustrates the effect of the cylindrical constraint.

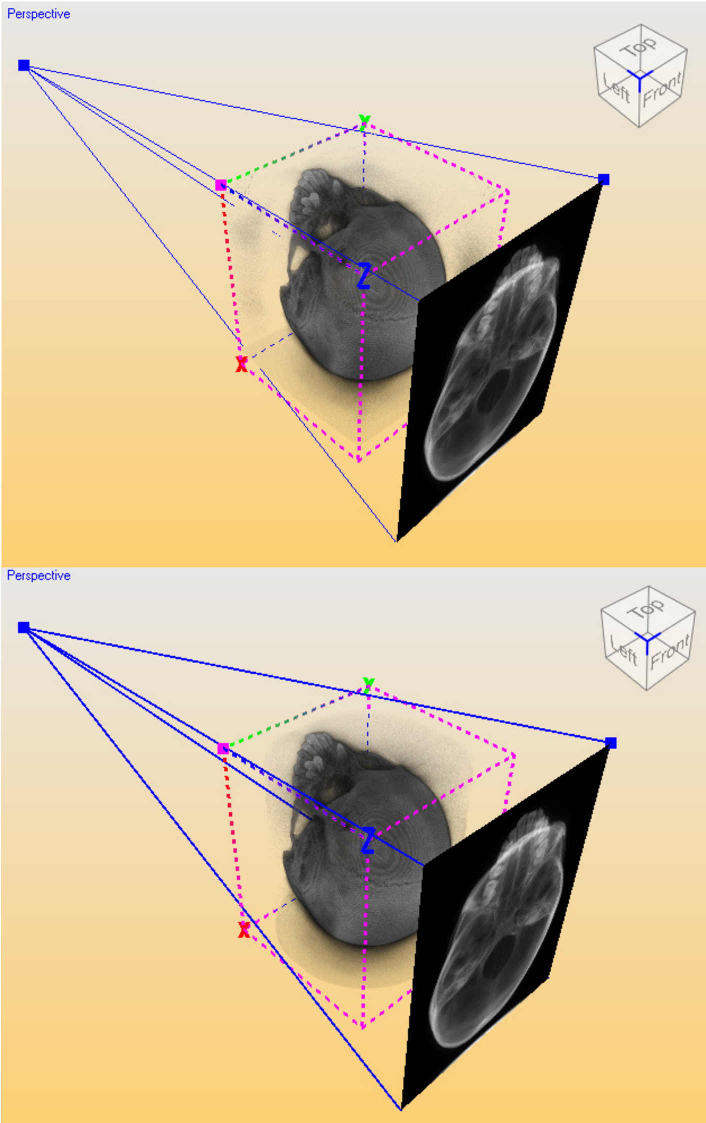


Figure 3.36: Effects of the cylindrical constraint on reconstruction. Top: without cylindrical constraint, bottom: with cylindrical constraint

3.6.6 Reconstruction grid orientation

In expert mode, it is possible to rotate the volume according to an orientation chosen during the preview step (see figure 3.37). As seen in the chapter 2.5, several axis references are used in the software. The modification of the quaternion acts directly on the "volume reference". To modify the orientation of the object, just fix the axis of rotation and the value of the angle of rotation.

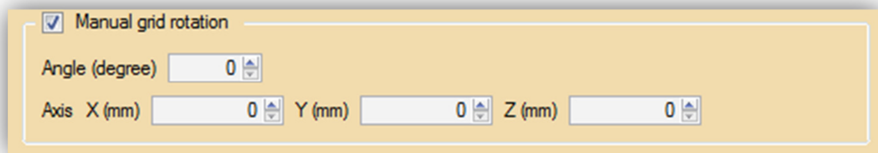


Figure 3.37: Rotation grid definition

During the modification of the grid rotation parameters, an overlapping box is added in which it can be seen the volume preview. This additional box illustrates the rotation that will be applied to the reconstruction grid with regard to the original reference.

3.7 Reconstruction [mandatory]

Digisens software allows you to select used reconstruction algorithms regarding to the geometry of the acquisition system or scan set up. Hence, you can choose among standard reconstruction methods such as FBP/FDK and iterative approached leading to improved image quality in strongly degraded context (low SNR, low contrast, limited view angles, few projections …).

To select any reconstruction algorithm, you just have to click on the arrow of the combo box to display the list of available methods (see figure 3.38).

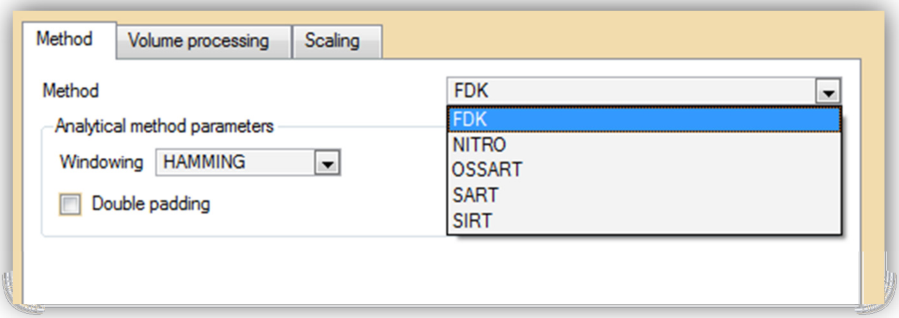


Figure 3.38: Reconstruction methods list

3.7.1 Filtered back projection

Filtered back projection is the most known approach in tomography reconstruction. Its ease of use and speed make it a reference in the field of application. This approach is a member of analytical reconstruction methods.

Regarding the beam type (See chapter 3.3.3), the list shown figure 3.16 is updated form FDK (cone beam) to FBP (parallel beam).

FDK or FBP need the filtering of projections by the mean of a ramp filter defined in the frequency domain. The main drawback of the ramp filter is that it amplifies noise as it restores object resolution. In order to reduce noise degradation in reconstructed volume, an apodisation filter is combined to ramp filter. Its aim is to reduce or to eliminate high frequencies in the image (noise). Unfortunately, high frequencies are necessary to restore the object resolution. Hence, filtering projections is a balance between signal to noise ratio and resolution. That is why the reconstruction module proposes several apodisation filters, each one making its own balance: ramp filter for high resolution reconstruction but with lower signal to noise ratio than a combination of a ramp filter and a Welch filter that increases signal to noise ratio but reducing resolution.

The figures below show the shape of combined filters in Fourier space. Hence, it can be seen that spare filter (ramp filter without apodisation) restore resolution regardless to signal to noise ratio. On the opposite,

Welch filter combined with the ramp filter increases signal to noise ratio regardless to resolution. Using those apodisation filters leads the user to accept a trade-off between noise level and resolution.

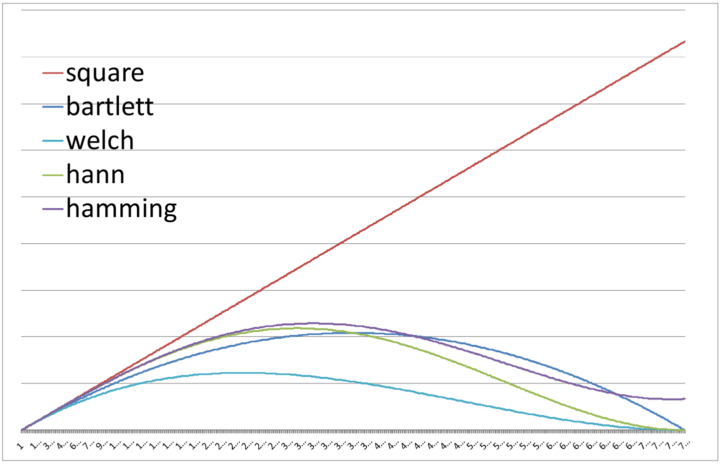


Figure 3.39: Reconstruction filter comparison

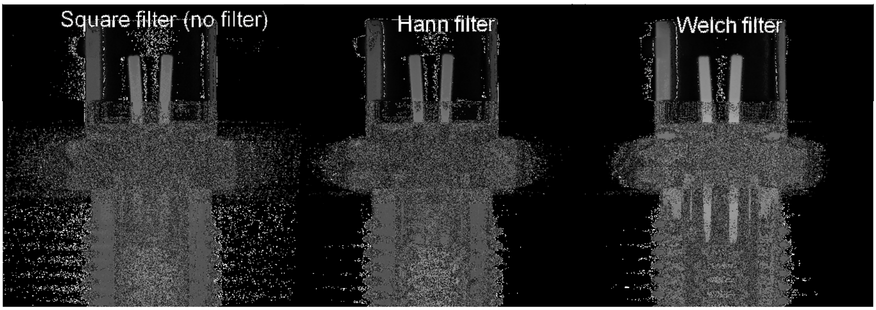


Figure 3.40: Examples of apodisation effects regarding noise level and reconstruction resolution.

3.7.2 Iterative algorithm

Despite its ease of use and speed, analytical approaches suffer from strong artifacts in case of degraded projections (low contrast, low SNR, …) and/or degraded scans (limited view angle, limited number of projections, …)

Iterative algorithms help in retrieving better image quality in such contexts. Unfortunately such approaches are not as easy as FBP/FDK to tune. That is why Digisens released NITRO® algorithm, a new iterative-based reconstruction approach.

3.7.2.1 NITRO® reconstruction method

Digisens developed NITRO® method for the following purposes:

- to provide the user with an easy to use tool
- to provide improved image quality with few noisy projections (see figure 3.44) or from strongly degraded acquisition set up
- to allow reconstruction of region of interest (see figures 3.45)

NITRO® is an iterative approach tuning automatically its initialization, its convergence speed and its regularization strength. The aim is to provide the best image quality whatever scan conditions.

Non-expert user will just need to select NITRO® method and to launch the reconstruction. For expert one, the tab `Volume Processing` will provide an access to the following parameters:

- Regularization from TV approach

Figure 3.41: TV regularization parameters

- Regularization from the number of iterations

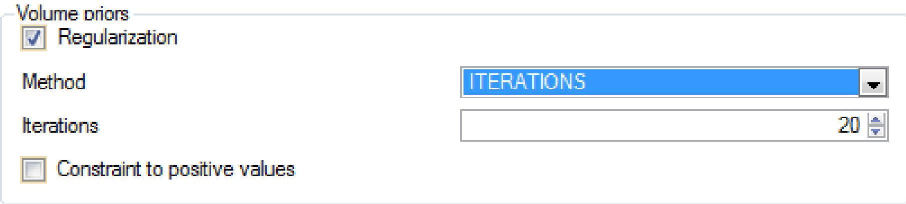


Figure 3.42: Regularization from the number of projections

- Positive constraint

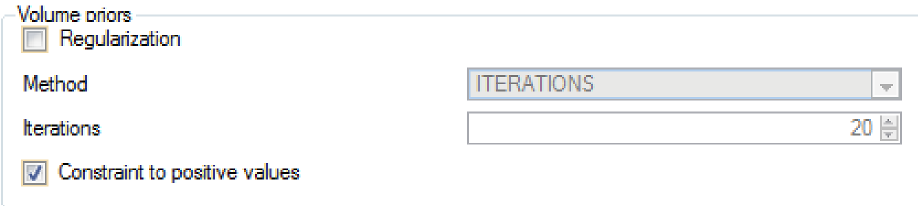


Figure 3.43: Positive constraint

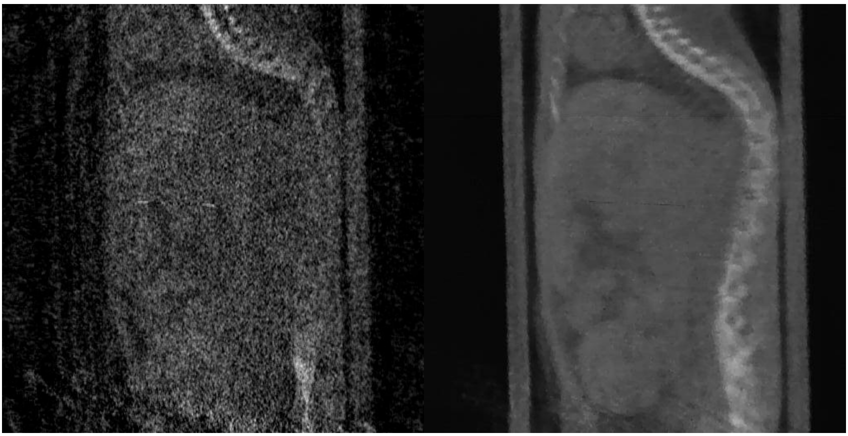


Figure 3.44: Example of the reconstruction of a mouse from 24 noisy projections

3.7.2.2 ART-based family

Digisens product also implements more standard methods based on Algorithm ART (Algebraic Reconstruction Techniques). This algebraic method is used in reconstruction of tomography data.

The algorithm implements the formula:

$$v_{k+1} = v_k + \lambda \frac{M_i^{\,t}(R_i - M_i v_k)}{\left\| M_i M_i^{\,t} \right\|}$$

With:

v_k current estimation of the volume

M_i Matrix of projection of the volume on the plan of the image sensor Ri and factor convergence λ.

ART formula is interpreted as follows: the error between the measures and the projection of the estimated volume is back-projected on the object space.

- An update of the volume is obtained by averaging the corrections of a set of projections (subset). If this set contains only one projection, then, the algorithm is a SART algorithm. If it contains all projections, then, the algorithm is a SIRT algorithm. The algorithm OSSART is a variant consisting in the launch of several SIRT with subsets containing a set of images lower than the total number of projections.

- The iteration number must be fixed by the user: as the low frequencies are reconstructed first, it is necessary to increase the iteration number to reconstruct the details of the object.

- The convergence parameter (convergence speed) must be in the interval] 0, 2[. It balances the correction which is applied to the current volume. It can be chosen manually or automatically (it depends of the subset size: Auto mode).

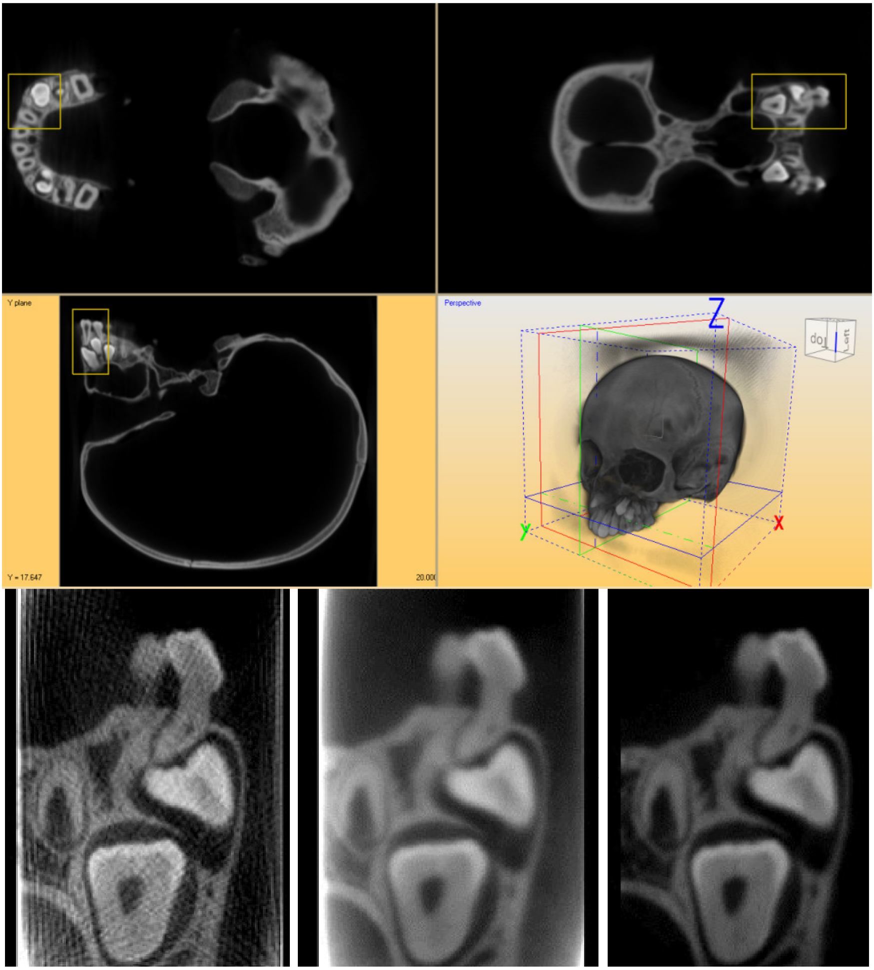


Figure 3.45: Example of the reconstruction of a region of interest of a human jaw: teeth close-up. Bottom left image is a reconstruction without SART algorithm, bottom middle image with SIRT algorithm, bottom right image with NITRO®. The two first algorithms have the well-known truncation artifact.

- The regularization depicts a minimization scheme (SNN filter or ICM minimization with a total criterion variation) during the

reconstruction, allowing the introduction of prior knowledge on the volume (ie piece wise constant objects). The effect of the regularization is balanced by the factor `Lambda` ($\in \left]0,1\right]$).

In comparison with analytical algorithms, iterative algorithms:

- Are more robust in relation to noise
- With fewer limited view angle artifacts
- Are slower (necessity to iterate)
- The whole volume needs to be stored in the memory

In practice, algorithm SIRT will privilege a good signal to noise ratio (SNR) regardless to resolution while algorithm SART will make the opposite. Algorithm OSSART will be between the both.

The ART mode is only available with GPU reconstruction. The multi-GPU reconstruction is possible if the volume can be stored on all GPUs available in the reconstruction system.

The particular case of the parallel geometry with vertical axis allows the use of multi-GPU without restrictions.

Important note: because of iterative specific implementation, the largest reconstructed volume corresponds to half of the total GPU memory, one third if 3D post processing is used.

Three reconstruction schemes are available in the list `Method` (see figure 3.38) :

- SART
- SIRT
- OSSART

If the OSSART is chosen, the user defines the value of the parameter `Projections Subset`.

It is possible to modify the value of the parameters `Iteration` and `Convergence Speed`.

The number of iterations and the value of Convergence Speed parameter can provide optimal convergence, whatever the method chosen (see figure 3.39).

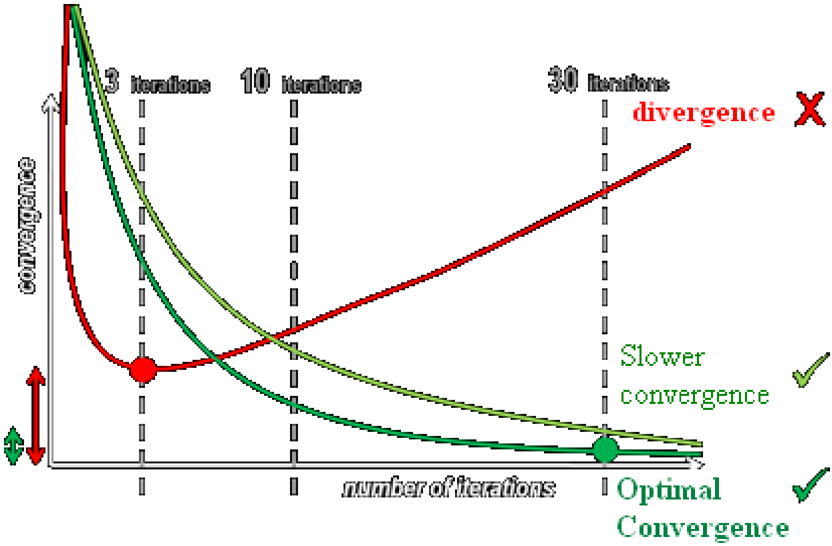


Figure 3.39: Iterative algorithm convergence

To improve image quality, the user can use post processing described in chapter 3.7.3, but also specific features for iterative algorithms. These features are:

- Regularization: the expert user can use regularization schemes into the process of reconstruction in order to increase the signal to noise ratio and to emphasize details of the reconstructed objects.

  He will have the choice between Symmetric Nearest Neighbor (SNN) and Total Variation (TV) (figure 3.40).

Figure 3.3 : Regularization parameters

The `Strength` parameter which is used as a weighting factor corresponds to the balance between prior knowledge and measurement.

Ideally, if its value is 100%, as much of weight to the regularization that to the data is granted, if its value is 50%, 2 times more importance with the data that with the regularization.

- Positive constraint: the option "`Constraint to positive values`" fixes all negative value in the reconstruction to 0 values.

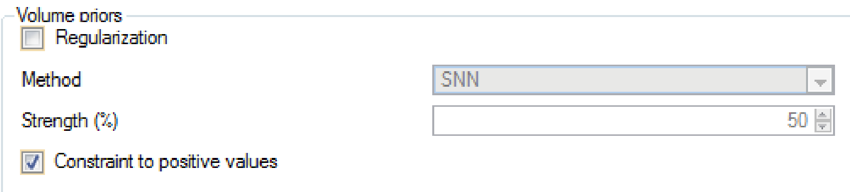


Figure 3.41: Regularization

3.7.3 Post-processing

You can apply to the reconstructed volume 3D post-processing to improve your image quality.

3.7.3.1 Ring artifacts remover

Ring artifacts have several origins. All of them are connected to an incoherence of pixels behavior with radiation. We can quote for example dead pixels, or pixels with the variability of their response to illumination with regard to the others. In this last case, the incoherence between the flat field image and the acquisition can be at the origin of ring artifact in the reconstruction.

Several approaches exist to correct these pixels. Several tools are proposed in pre-processing of the projections or the sinogram itself. The tool proposed here is applied to the reconstructed volume. The principle is based on the detection of rings and their deletion.

Both accessible parameters in expert mode are the maximal thickness of rings to be deleted (`size`), and `Alpha` which takes into account if rings are complete and constant or modulated (ie partial rings).

By default, parameters value is 3 for `size` and 0.2 for `Alpha`.

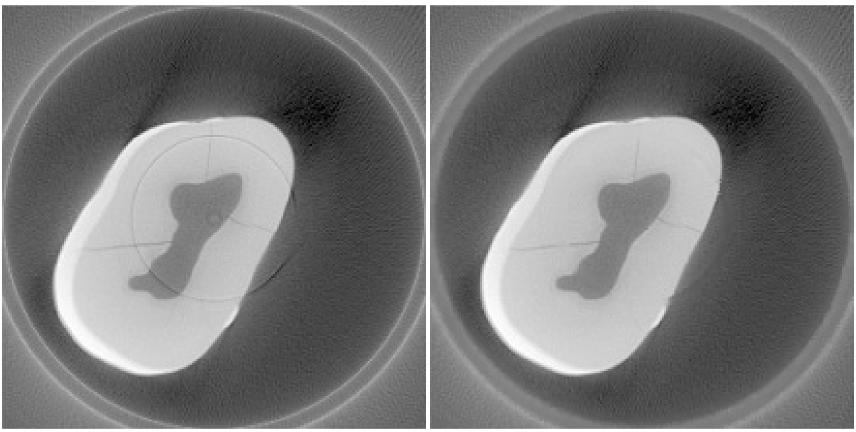


Figure 3.42: Ring artifact reduction. Left image: raw image with ring artifacts. Right image: filtered image.

Important note: Ring artifact filter works only if the object is in its natural orientation (vertical axis by default merged with the axis of rotation). If the

grid orientation is used to rotate the object after the reconstruction, this filtering is not functional. The user will be warned of it by a message in the screen. If however the rotation is compulsory, it will be possible after reconstruction to rotate the object by using the reference axis modification described in the chapter 2.5.2.

Important note: Ring artifact filter is not available in slice preview mode.

3.7.3.2 3D edge preserving filter

This filter is a 3D extension of the filter described in 3.4.2.3.

3.7.3.3 Bilateral filter

Bilateral filter is an edge-preserving noise reducer. In homogeneous regions, a Gaussian kernel is used to smooth pixel intensity in a range defined by kernel size. Weighting function is composed of two Gaussian kernel:

- A first kernel based on distance from processed pixel, defined by its standard deviation σ_{xyz}
- A second kernel based on pixel intensity, describing intensity gap defining edges. The kernel is defined by its standard deviation σ_g

Figure 3.43 shows how the filter works: left image shows the signal to process, middle image the kernel shape and right image the resulting processed signal.

To use this filter (see figure 3.44), check the bilateral checkbox and fix the two filter parameters: one for intensity and one for spatial characteristics.

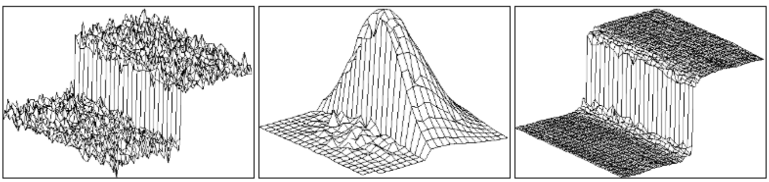


Figure 3.43: Kernel definition.

Note that those parameters are not absolute values, meaning that S_g corresponds to intensity standard deviation relative to the min-max range of grey levels in the reconstruction. Its max value is consequently 1. This is the same for S_{xyz} but regarding volume resolution.

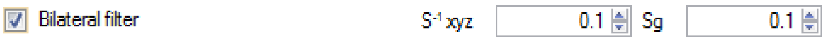


Figure 3.44: Bilateral filter parameters

The implementation of this filter is fast, but the speed is in straight relation with parameters value.

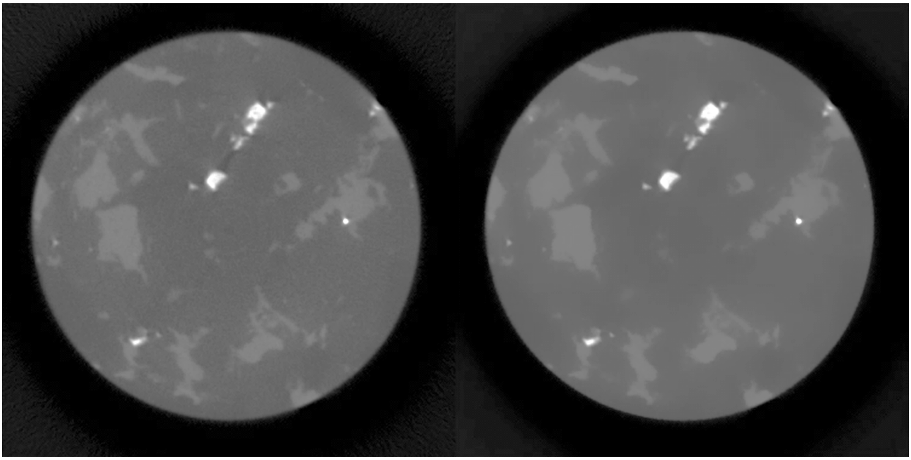


Figure 3.45: Effects of bilateral filter (reconstructed slice of a granite sample)

3.7.3.4 Sharpness filter

In tomography application, system resolution is usually degraded by focus spot finite size, detector pixels and scintillator, reconstruction algorithm… This degradation can be seen as a blurring kernel making reconstruction edges smoothed after reconstruction process. To avoid such degradation, Digisens has implemented a filter able to retrieve sharp edges.

To use this filter (see figure 3.46), check the `sharpness filter` checkbox and fix the three filter parameters:

- Number of iterations used for processing
- A radius corresponding to a low pass filtering necessary to make filtering stable.
- A speed parameter corresponding to a discretization step. If the value is decreased, filtering is slower but finer

Figure 3.46: Sharpness filter parameters

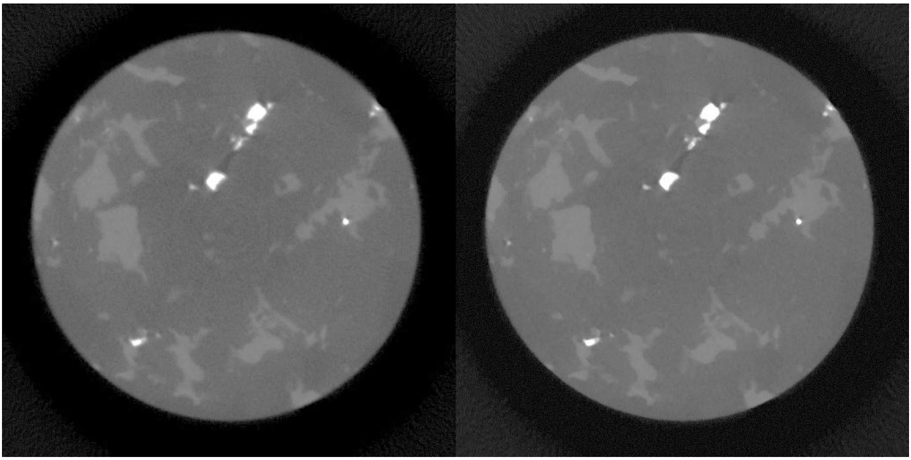


Figure 3.47: Effects of sharpness filter (reconstructed slice of a granite sample)

3.8 Preview and Adjustment

3.8.1 Preview Settings

Preview reconstruction allows the parameters entered in previous stages to be quickly checked, and, possibly, to be adjusted iteratively (ie

adjusting the trajectory definition, or even, the dimensions of the volume to be reconstructed).

DigiR3D proposes two preview modes:

1. reconstruction of the whole 3D object in low resolution;
2. reconstruction of a single high resolution slice which is arbitrary placed thanks to the cutting plane.

The two modes pre-calculate a sub collection of the low-resolution filtered projections in low resolution to accelerate the successive previews: it is this sub collection that we refer to as “cache”.

It is possible in the `Images for preview` of the dialogue box to modify the number of projections to be used as well as their resolution (by default adapted to the size of the low resolution volume). The limit which prevents the use of all of the projections in full resolution is the memory used by this cache which is displayed in `memory used`, for information.

Low resolution volume preview 

Low resolution preview is accessible whichever tab is displayed at the bottom of the dialogue box. To launch the reconstruction, one has just to click on the button .

It reconstructs the corresponding volume in a few seconds. By default, three resolutions are offered: 128^3, 256^3 and 500^3. Thus, we have a visualization which is close to that which will be reconstructed (see figure 3.48).

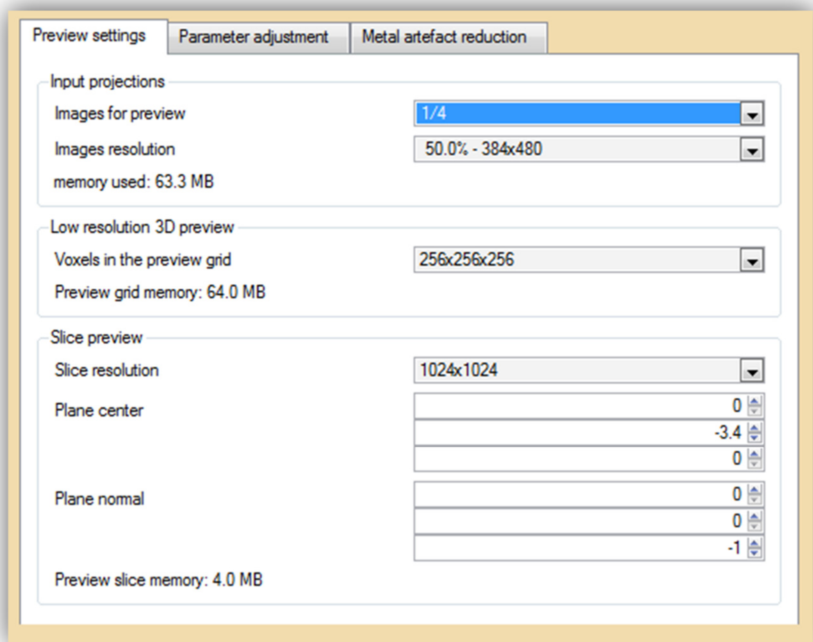


Figure 3.48: Preview parameters

It is then possible to quickly check the coherence of the entered parameters, and to adjust the volume reconstructed on a possible area of interest.

Note: It is possible to manually enter a reconstruction resolution, such as for example “32x128x64”. However, as the reconstructed volume is only destined for 3D display, we recommend that you follow the restrictions imposed by the graphics cards used: the resolutions should be a power of two, and for most graphics cards, the 3D display resolution is limited to 512. Not following these restrictions does not involve an error, but causes re-sampling (useless) for the display with respect for these restrictions.

Note: volume preview remains displayed even after the closure of the dialogue box. It is possible to remove it from display by un-ticking the `show Preview` box.

Preview by high resolution slice 

Preview by slice is only available if the ART reconstruction is not selected in Preferences.

Preview by slice is accessible regardless of the tab displayed at the bottom of the dialogue box. To launch the reconstruction, one has just to click on the button .

It allows the reconstruction of a slice arbitrarily placed in a greater resolution (1024x1024 by default).

The position and orientation of this slice can be set in the dialogue box by specifying a `Plane center` point and a normal to this plane (`Plane normal)`. It can also be set by the normal cutting plane which can be manipulated in 3D display, for example on low resolution volume preview.

Note: The successive slice previews open a new display window each time to allow comparisons between the different slices/versions.

Note: It is possible to interactively edit the title of the window to put a more appropriate comment in it on the parameters being modified, for example. By default, the title re-uses the position of the calculated slice.

In following figure, we can see two 512x512 slices calculated with two sub-collections of different radiographs.

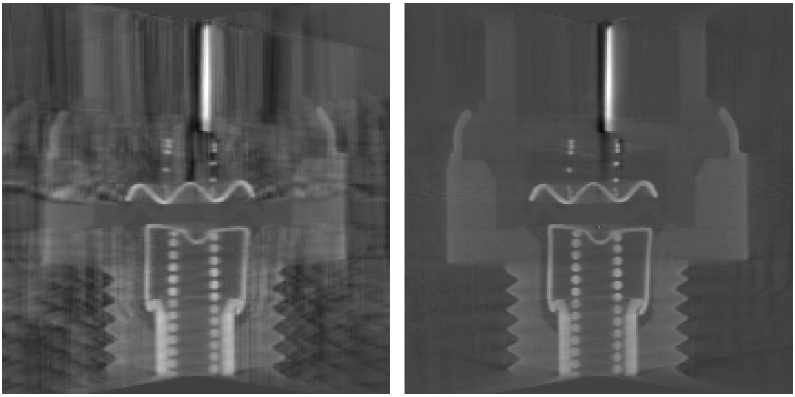


Figure 3.49: Preview of a 512x512 pixel slice: with 30 128x160 pixel radiographs on the left and 180 768x960 radiographs on the right.

This illustrates the importance of the number of radiographs used and their resolution on the quality of previews. The slice on the left has been calculated based on 45 images of 128x160 pixels (reusing the same cache as the precedent volume preview). The slice on the right has been calculated based on 180 images (one out of two) of 768x960 pixels (full resolution).

3.8.2 Adjust

Although geometric parameters or beam hardening coefficient can be finely adjusted thru the previews, in practice, it can turn out to be tedious to wait for the end of the calculations during multiple iterations on a given value. It can also be impossible to get a sufficiently accurate preview, particularly because of memory limitation which prevents the use of all the full resolution radiographs available in the preview cache.

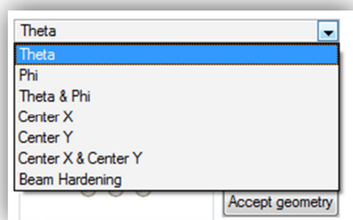


Figure 3.50: Parameters list

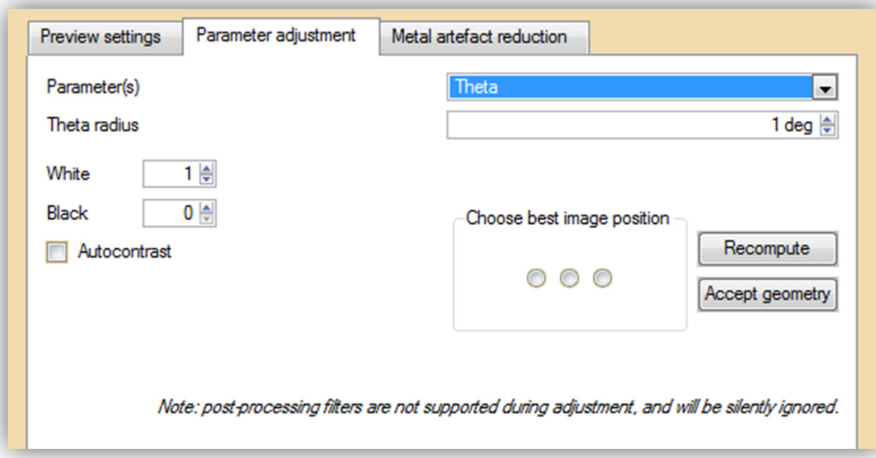


Figure 3.51 Adjustment tab GUI

For these two reasons, a semi-automatic tool is proposed (see figure 3.51). The user chooses on which variable he wants to make the adjustments. Then, the parameter increment has to be set up (`step`). By clicking on `Recompute` button, the software computes the slices defined in `Preview Settings` tab with the following parameter values:

- `Init_value - increment`
- `Init_value`
- `Init_value + increment`

Resulting slices are displayed in the main window.

Note: for the first step, the initial value used corresponds to the one set up in the geometry definition tab.

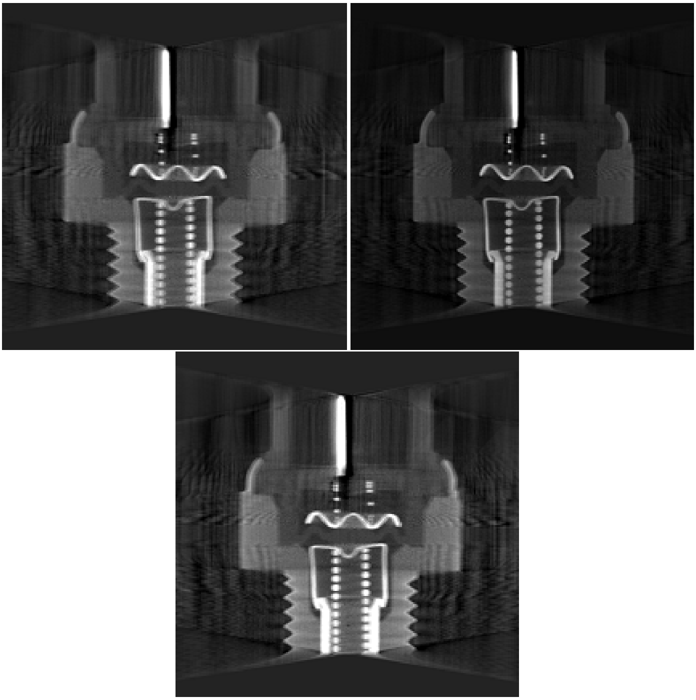


Figure 3.52: Examples of "Z" slices corresponding to the center of rotation positions (-0.05,0,0), (0,0,0) et (0.05,0,0) from left to right.

When processing is complete, to inform which new value to take, one has to click on the radio button corresponding to the best reconstructed slice. The aim is to set the best image in the center of the different views and then click on `Accept geometry` button to update the geometry dialog tab.

3.8.3 Metal artifact reduction

Metal artifact is a well-known artifact in reconstruction of complex parts with very high disparity in density level. This artifact can degrade reconstruction in such a way that no analysis is possible (see figure 3.53): information signal around the higher density material is degraded by streak artifact or a 0 density area (information lost).

This artifact has two origins:

- Beam hardening (see chapter 3.4.2.4)
- Photon starvation

DigiR3D proposes to the user a tool dedicated to the restoration of information in the area surrounding the high density area.

To do so, please follow the steps:

- Check Use metal artifact reduction checkbox
- Select an initial threshold value >>1
- Click on `Compute slice` to compute a slice preview
- Than modifying the threshold value will paint pixels associated to high density value in red (see figure 3.54).

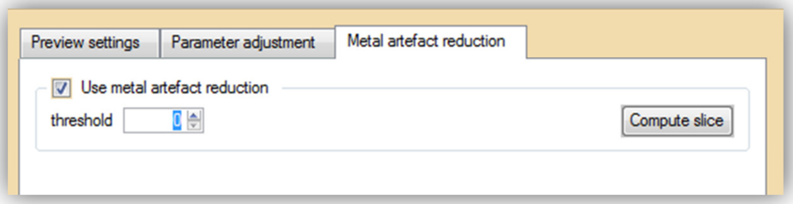


Figure 3.53: Metal artifact reduction GUI

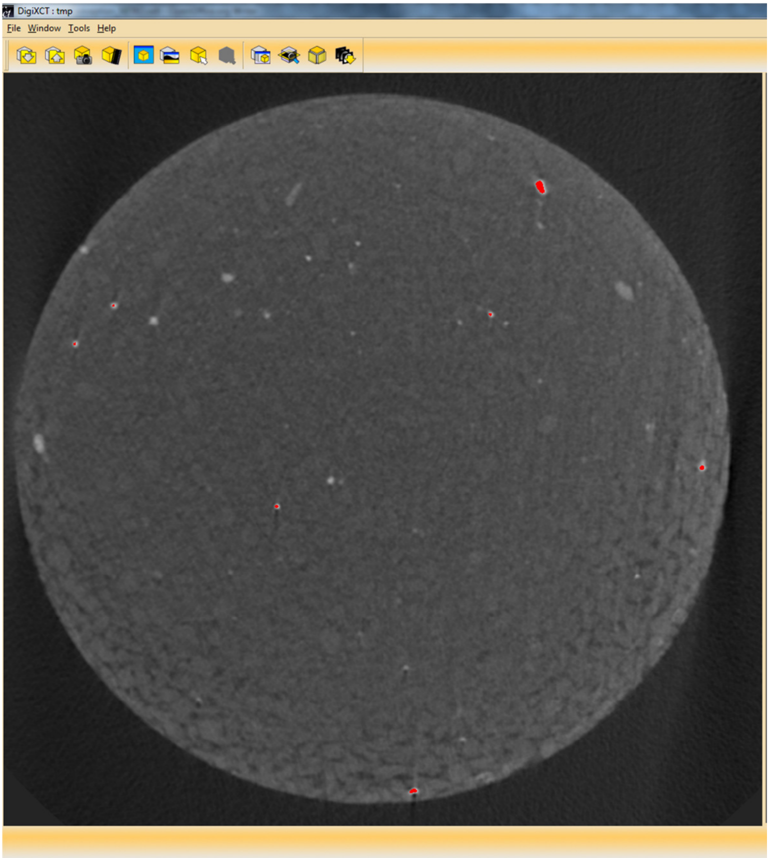


Figure 3.54: High density material segmentation

-When the segmentation of high density pixels is good enough for the user, compute a slice preview using  or a 3D preview using  to check the result.

If threshold value is validated, launch the reconstruction, if not refine the threshold. Figure 3.55 shows the reconstruction after metal artifact compensation.

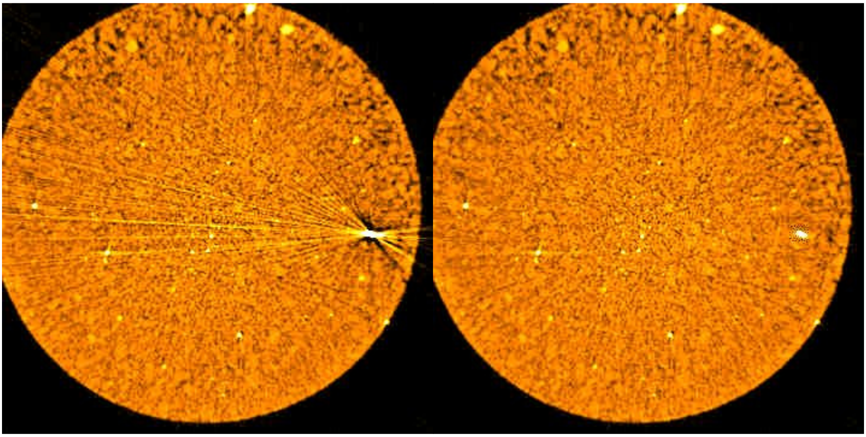


Figure 3.55: Reconstructed slices: left image without MAR correction, right image with MAR correction.

3.9 Save, load and reconstruction

The `load` and `save` buttons  allow the reading and the writing of all of the reconstruction parameters in a calibration file in a proprietary format of which the extension is.cal.

When you click on the reconstruction button , a dialogue box summarizes the defined parameters for a final validation with the effective launch of the calculations.

4 Analysis of cutting planes: DigiCUT plug-in

As with all the plug-ins in the DigiXCT software suite, DigiCUT is protected by a software key.

4.1 General presentation of DigiCUT

The `Cutting plane Analyzer` command ( icon in the plug-ins' button bar or `[Tools → Cutting plane analyzer]` in the menu) allow a 2D cutting plane to be visualized and analyzed (Figure 4.4).

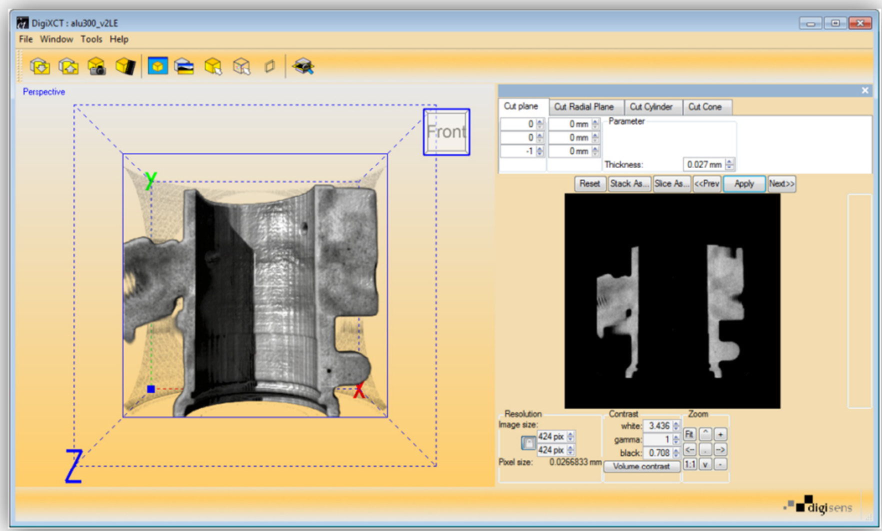


Figure 4.4 : Cutting plane analysis dialogue box

4.2 DigiCUT functionalities

4.2.1 Resolution

`Image size` and `pixel size` are set into `Resolution` area. It allows the modification of pixel dimensions in millimeter and image resolution in pixels.

4.2.2 Contrast

`Contrast` can be set using `white / black / gamma` buttons. `Volume contrast` button fixes the image contrast using 3D volume histogram range.

4.2.3 Moving in the image

The `Zoom` zone is composed of 9 buttons:

- ✓ Button 'Fit' : Fixes the size of the image to the window size
- ✓ Button '1:1' : Fixes the size to the scale of 1:1 (1 image pixel corresponds to 1 screen pixel)
- ✓ Button '.' : Centers the image in the window
- ✓ 4 arrows : Moves the image in the window
- ✓ Button '+/-' : Zoom +/-

It is also possible to zoom in on a precise rectangle by holding down the `shift` key and then clicking the left mouse button, by moving, then releasing the button.

4.2.4 Other features

The `help` button opens the help window for the different commands as well as the short cuts. The `Save image` button allows the current image to be saved. The `close` button closes the window.

A right click on the slice image allows a contextual menu to appear. The menu has the following commands:

- ✓ `Help` : Displays the help window
- ✓ `Save image` : Saves the image
- ✓ `Level curve mode` : Activates the density profile

examination mode

✓ `Mesure mode` : Active le mode de mesure

4.2.4.1 Level curve mode

During the analysis of a density profile, the line being analyzed is marked by a green line (Figure 4.2). A left mouse click allows it to be moved. It is possible to switch the analysis with a horizontal line or a vertical line by ticking the *vertical* box in the contextual menu or not. The data can be exported in Excel format with the `Export curve data` command in the contextual menu.

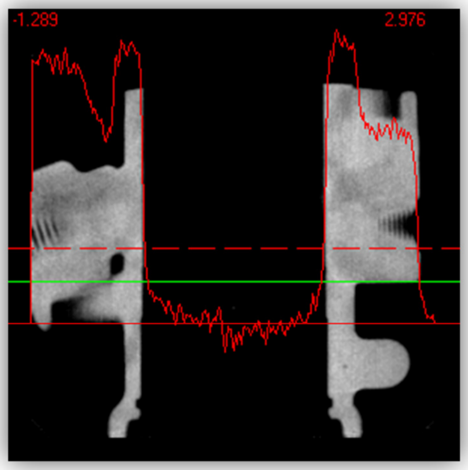


Figure 4.5 : Example of density profile analysis

4.2.4.2 Measures

Activation occurs by right clicking and selecting `Measure Mode`. The activation provokes the appearance of a bar of buttons offering multiple types of measures (figure 4.3).

Important note: this tool proposes informative measure, but it does not claim to give a so precise measure as a metrology software would make it. The expected precision here will be in relation with the position of the

mouse click and to the resolution of the image. We shall thus have a precision in the pixel. That is the notion of informative measure.

Note: at any moments it is possible to refine the position of the selected points.

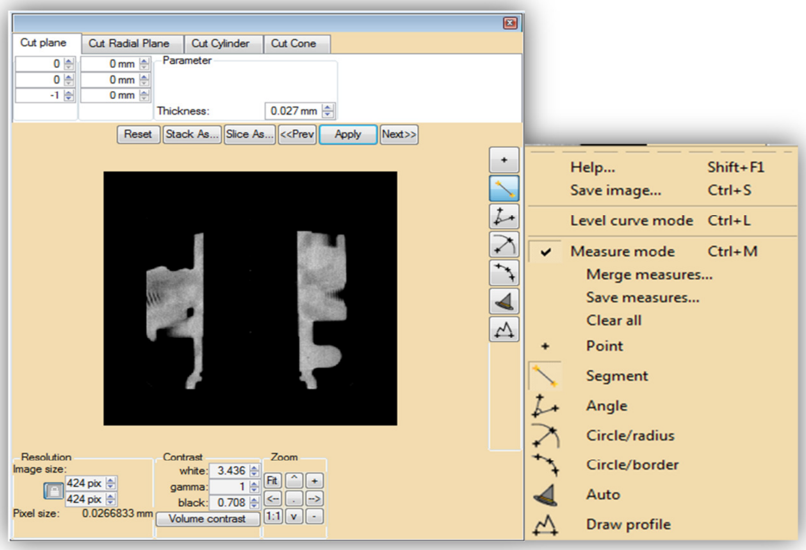


Figure 4.6 : Different measures proposed

4.2.4.2.1 *Measure reference*

All necessary information about reference can be found in chapter 2.5.2.

4.2.4.2.2 *Point*

To select a point in the image, you need to:

- Select point measure
- Left-click on desired pixel in the image

Displayed information is:

- X,Y,Z location in measure reference
- Corresponding grey level in the pixel

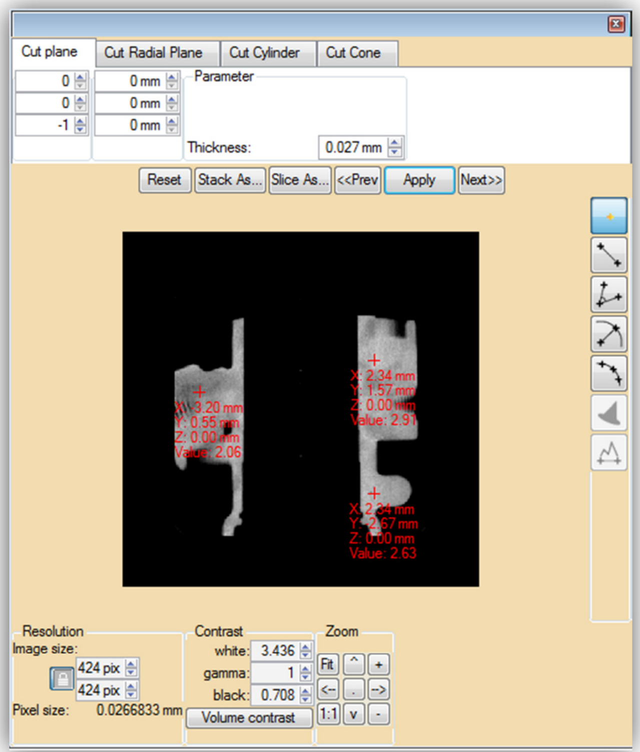
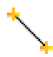


Figure 4.4: Points measure

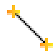
4.2.4.2.3 Length measure

To measure a length, two methods are available:

- First method :
    - Select segment measure 

- Verify that the assisting mode is deactivated 
- Left-click with the mouse to select the origin point of the segment
- Left-click with the mouse to select the ending-point of the segment

Segment limits are represented with small squares.

- Second method :
  - Select segment measure 
  - Select the assisting mode 
  - Left-click in the neighborhood of the desired origin point of the segment
  - Left-click in the neighborhood of the desired ending-point of the segment

The assisting mode integrates an edge detection tool based on the analysis of grey-levels along the profile drawn by the segment. Segment extremities are then adjusted to fit the real borders of the part one wants to measure. Les extrémités du segment s’ajustent alors au plus près du contour de l’objet affinant ainsi la mesure. Segment limits are represented with small vertical bars.

Displayed information is:
- Segment length

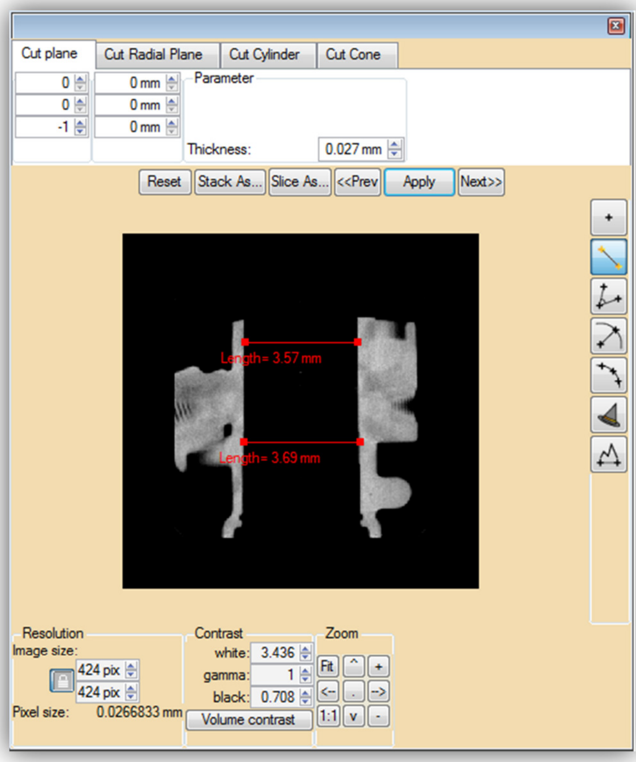


Figure 4.5: Segment length measure

4.2.4.2.4 Angle measure

To select an angle in the image, you need to:

- Select angle measure 
- Left-click to select the first point of angle definition
- Left-click to select the second point of angle definition
- Left-click to select the third point of angle definition

Displayed information is:

- Angle value between the two segments defined by first/second points selected and second/third points selected.

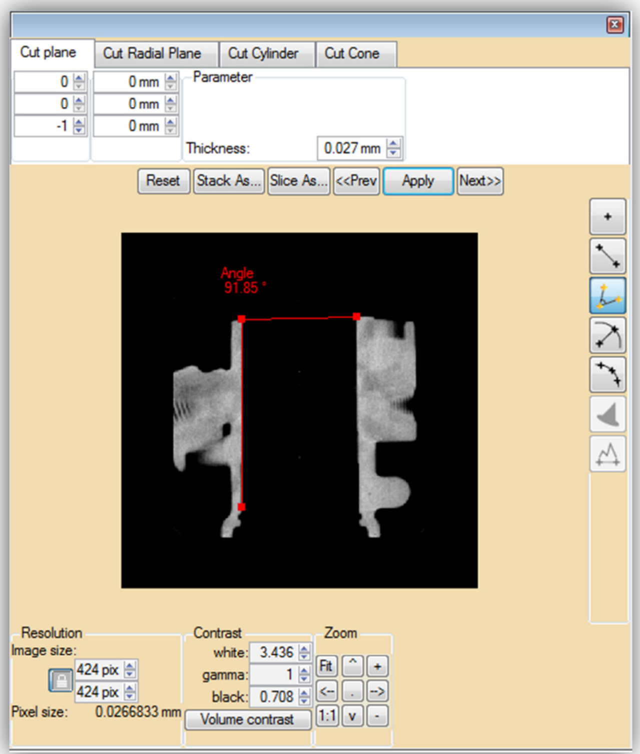
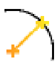


Figure 4.6: Angle measure

4.2.4.2.5 Circle measure

To measure a circle, two methods are available:

- First method:
    - Select circle measure 
    - Verify that the assisting mode is deactivated 

- Left-click to select the circle center
- Left-click to determine circle radius

Displayed information is:

- Circle diameter
- Location of the center in the reference
- Perimeter value
- Disk area delimited by the circle

- Second method :
  - Select circle measure 
  - Select the assisting mode  (it is possible to deactivate it)
  - Left click on 10 points defining roughly the circle you want to define
  - When the tenth point is selected, the circle is adjusted to fit the object border

Displayed information is:

- Circle diameter
- Location of the center in the reference
- Perimeter value
- Disk area delimited by the circle

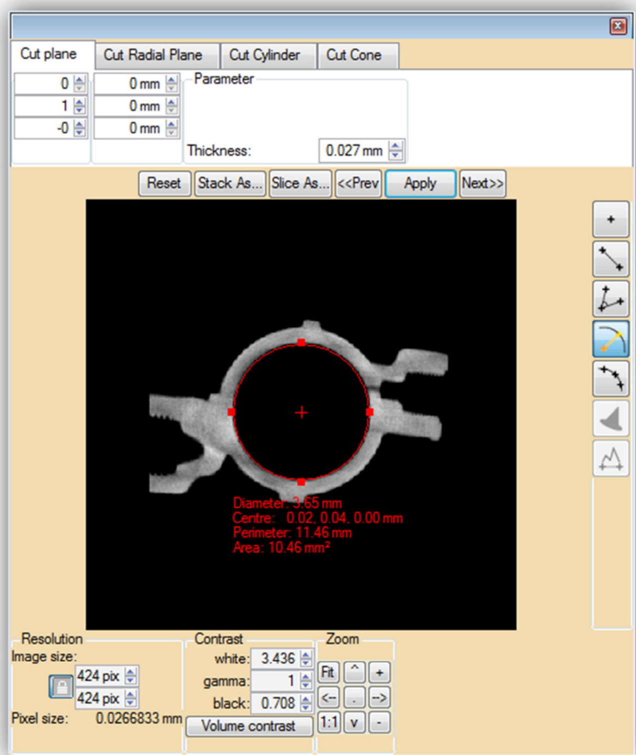
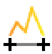


Figure 4.7: Circle measure

4.2.4.2.6 Profile measure

To measure a profile along a segment, you need to:

- Select segment measure
- Select profile edition 
- Measure the segment

The following figure illustrates the profile edition.

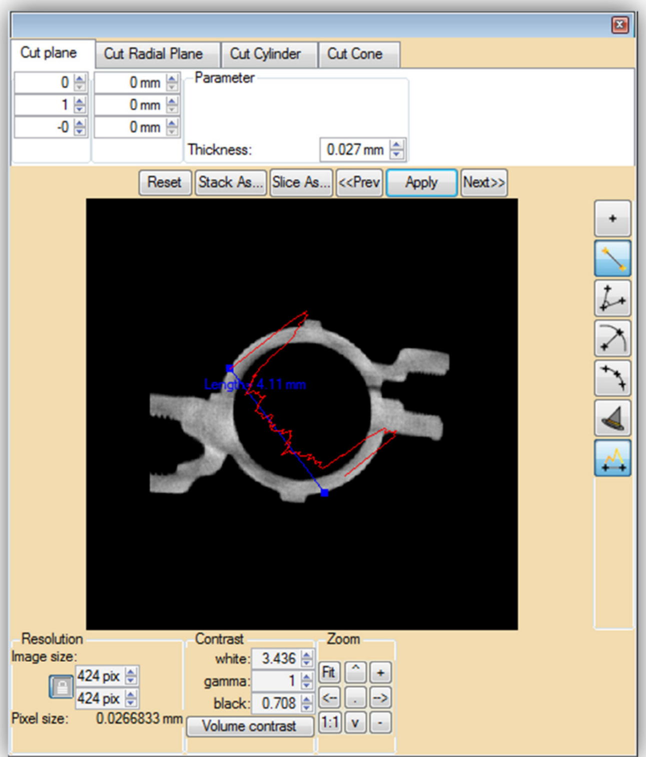


Figure 4.8: Profile measure

4.2.4.2.7 Other functionalities

Properties edition of the measure is made by clicking with the right button on the concerned point and selecting Properties. The dialogue represented on figure 4.9 appears. It is possible to modify the color of the measure (`Color`), the size of the text (`Text Size`), the number of decimals (`Decimals`), widths of the lines of measure (`Width`).

Right-clicking on a point also allows you to remove it or to remove the measure.

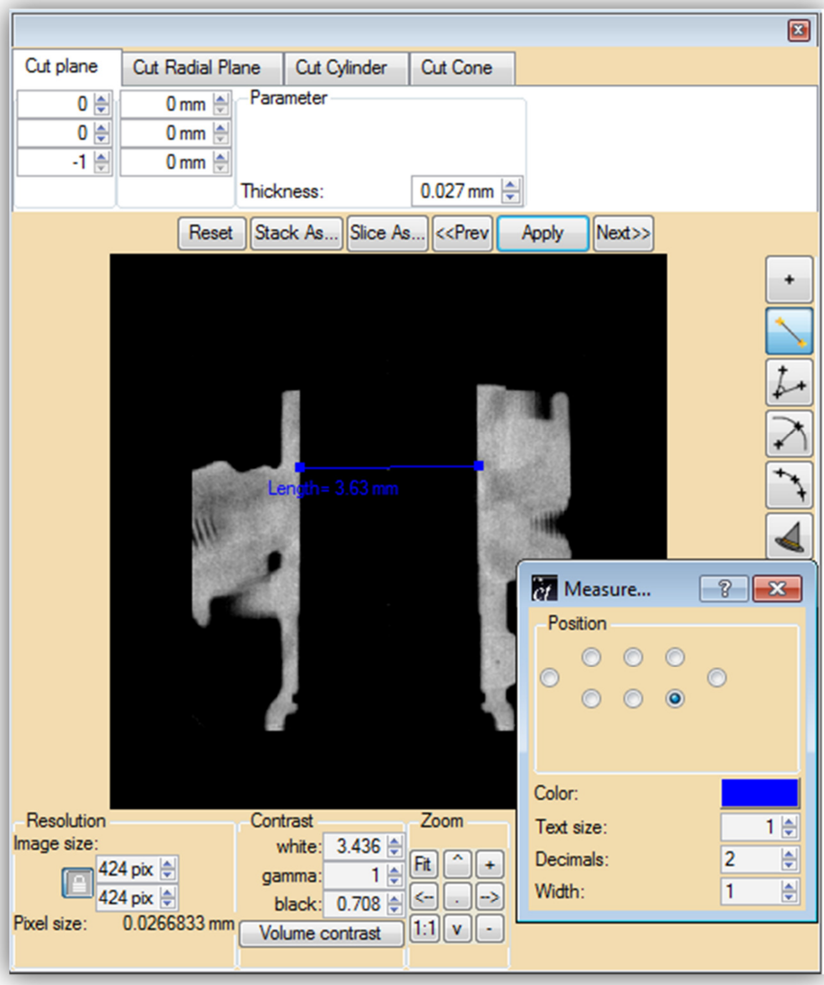


Figure 4.9: Measure example

4.3 Specific slice extraction modules

DigiCUT plugin manages with several extraction types: planes or curvilinear surfaces (cylinders or cones).

4.3.1 Cut Plane

`Cut Plane` tab computes a 2D slice image from 3D object using cutting planes. The `Apply` button computes the current cutting plane.

To locate the cutting plane, the user can fix the values of the plane normal and plane center filling `Normal` and `Center` boxes. The user can also move the clipping plane in the 3D view using the mouse in clipping plane management mode

Buttons <<*`Back`* and *`Next>>`* makes the cutting plane moves along the normal axes direction with a moving step equal to `Thickness`. Views are automatically recomputed.

Button *`Stack As`* grabs a set of images from the top of the volume to its end, along the normal axis direction.

Note: The step between two slices is fixed by the value *`Thickness`*.

Button *`Slice As`* grabs only the current displayed slice.

4.3.2 Cut Cylinder

This mode is an option of DigiCUT. Please contact Digisens for further information.

When `Cut Cylinder` tab is select (Figure 4.10), a transparent cylinder and its axis of revolution are displayed in the 3D view window. This superimposed object displays the curvilinear surface used to perform slice extraction.

There are two ways to change the cylinder orientation and the base: either axis and base coordinates are entered in the corresponding fields, or the user uses the mouse. To use the mouse, click on  or press the space bar and use the mouse as follows:

- ✓ *Right click* : Cylinder axis shift
- ✓ *Left click* : Cylinder axis orientation change

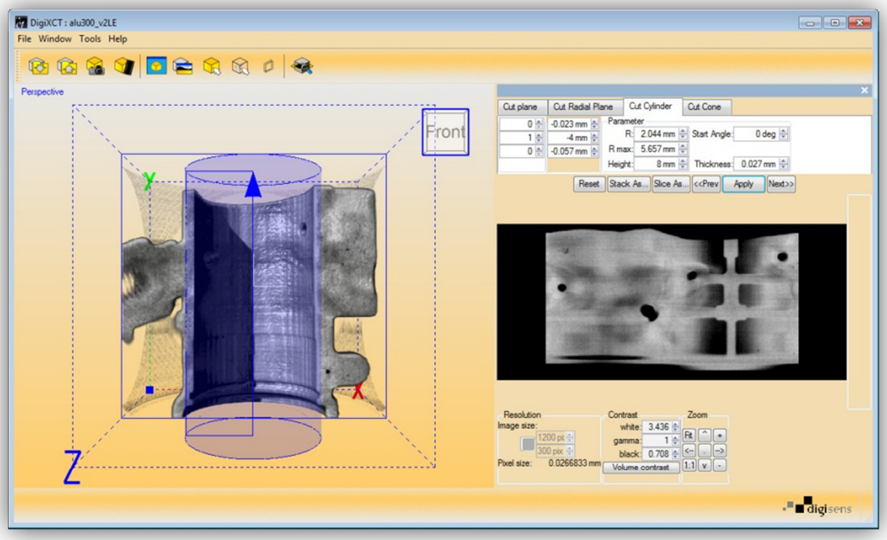


Figure 4.10: Cut Cylinder module

To extract a slice, first fix the following values:

- Base radius `R`
- Cylinder height `Height`
- Revolution axis `Axis`
- Cylinder base origin `Base`
- Maximum radius for stack extraction `Rmax`

When parameters are fixed, click on `Apply` button to get the corresponding slice.

Note: regarding object dimensions, the computation may take several seconds.

Buttons <<`Back` and `Next>>` makes the cutting plane moves along the normal axes direction with a moving step equal to `Thickness`. Views are automatically recomputed.

Button `Stack As` grabs a set of images. Each image corresponds to a cylinder base radius increased by a radius step equal to `Thickness`. The base radius range stats at `R` and finishes at `Rmax`.

Button `Slice As` grabs only the current displayed slice.

Note: the value `Rmax` is used to fix slice image dimensions.

The user can set the origin of the image (first pixel) by fixing the `Start Angle` value. This value acts like an offset (from 0 to 360 degrees) along the perimeter of the base.

The `Reset` button relocates the cylinder axis and base to initial values.

Note: in this mode, the image size is automatically computed and is not editable.

4.3.3 Cut Cone

This mode is an option of DigiCUT. Please contact Digisens for further information.

When `Cut Cone` tab is selected (Figure 4.11), a transparent cone and its axis of revolution are displayed in the 3D view window. This superimposed object displays the curvilinear surface used to perform slice extraction.

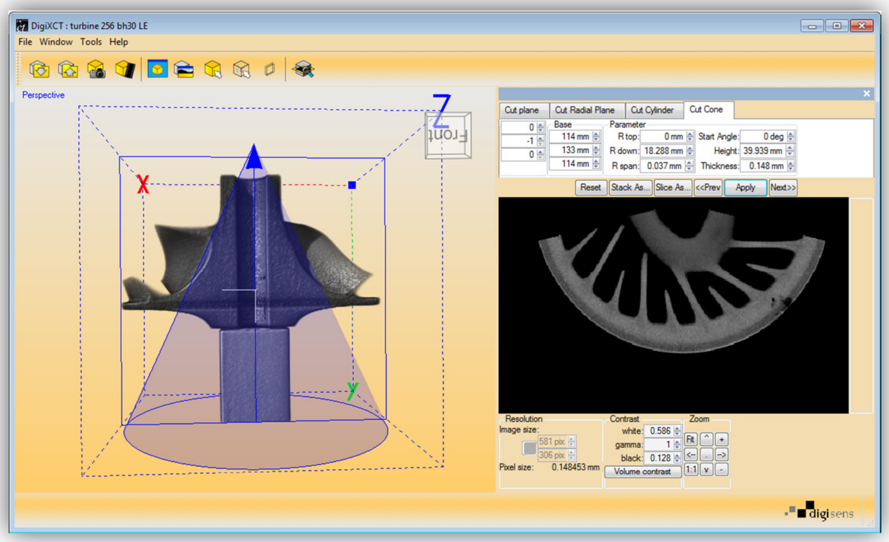


Figure 4.11: Cut Cone module

There are two ways to change the cone orientation and the base: either axis and base coordinates are entered in the corresponding fields, or the user uses the mouse. To use the mouse, click on  or press the space bar and use the mouse as follows:

- ✓ *Right click* : Cone axis shift
- ✓ *Left click* : Cone axis orientation change

To extract a slice, first fix the following values:

- Bottom base radius `Rdown`
- Top base radius `Rtop`
- Cone height `Height`
- Revolution axis `Axis`
- Cone base origin `Base`

When parameters are fixed, click on `Apply` button to get the corresponding slice.

Note: regarding object dimensions, the computation may takes several seconds.

Buttons <<`Back` and `Next>>` makes the cutting plane moves along the normal axes direction with a moving step equal to `Thickness`. Views are automatically recomputed.

Button `Stack As` grabs a set of images with cone-based surfaces which base radius ranges from [`Rdown – Rdown+RSpan`]. Each image corresponds to a cone base radius increased by a radius step equal to `Thickness`.

Button `Slice As` grabs only the current displayed slice.

For a truncated cone, the radius *`Rtop`* is equal to 0. Then, the height is computed in such a way the slope of each cone is parallel. The figure 4.12 depicts the two cone types used for extraction.

The user can set the origin of the image (first pixel) by fixing the `Start Angle` value. This value acts like an offset (from 0 to 360 degrees) along the perimeter of the base.

The `Reset` button relocates the cylinder axis and base to initial values.

Note: in this mode, the image size is automatically computed and is not editable.

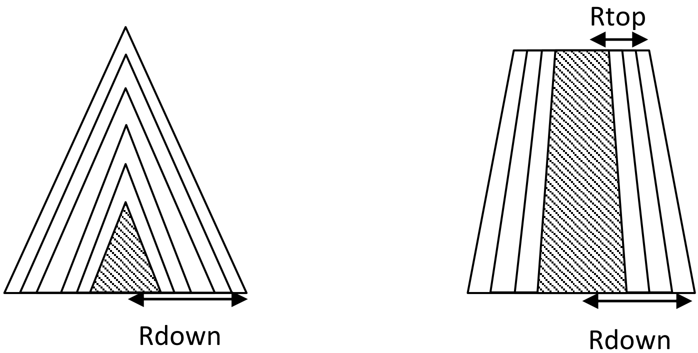


Figure 4.12 Stack slice representation in cut cone mode

4.3.4 Cut Radial Plane

This mode is an option of DigiCUT. Please contact Digisens for further information.

When `Cut Radial` tab is selected (Figure 4.13), a transparent frame and its axis of revolution are displayed in the 3D view window. This superimposed object displays the plane used to perform slice extraction.

There are two ways to change the plane orientation and the location: either axis and base coordinates are entered in the corresponding fields, or the user uses the mouse. To use the mouse, click on  or press the space bar and use the mouse as follows:

- ✓ *Right click* : Plane axis shift
- ✓ *Left click* : Plane axis orientation change

To extract a radial cut, first fix the following values:

- Frame width `Width`
- Frame height `Height`
- Revolution axis `Axis`
- Frame axis base `Base`
- Start angle for radial cut `Start Angle`

When parameters are fixed, click on `Apply` button to get the corresponding slice.

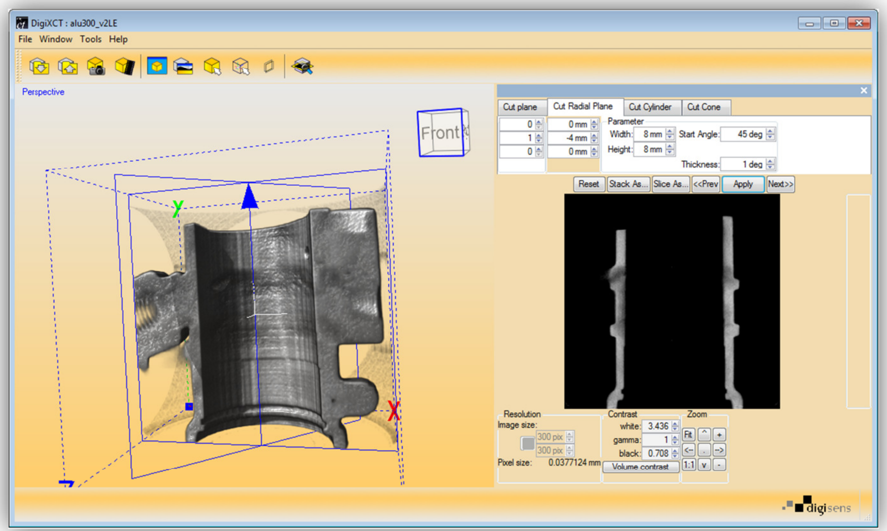


Figure 4.13: Mode coupe radial: Cut Radial Plane

Buttons <<`Back` and `Next>>` makes the cutting plane moves around its axis with an angular step equal to `Thickness`. Views are automatically recomputed.

Button `Stack As` grabs a set of images with a rotation range of 180 degrees. Each image corresponds to a cone base radius increased by a radius step equal to `Thickness`.

The user can set the origin of the extraction (first image) by fixing the `Start Angle` value (from 0 to 360 degrees). The `Reset` button relocates the cylinder axis and base to initial values.

5 Visualization and extraction of surfaces: DigiSRF plug-in

As with all the plug-ins in the DigiXCT software suite, DigiSRF is protected by a software key.

5.1 General presentation of DigiSRF

SRF is the DigiXCT software suite's surface extraction plug-in.

SRF calculates a surface corresponding to a given level of density based on a volume coming from a tomographical reconstruction. It is based on the *Marching Cubes* algorithm. This plug-in appears in the form of a dialogue box which is accessible via the `[Tools → Surfaces]` menu or via the button  .

5.2 DigiSRF functionalities

5.2.1 Surface manager

SRF's main dialogue box (Figure 5.1) allows surfaces to be created and removed, the volume visualization to be masked and the global parameters concerning optimization of calculated surfaces to be adjusted.

The `New` button allows a new surface to be created. It opens a new tab (see Figure 5.1). By default, the new surfaces are empty and nothing is displayed on the 3D display. Refer to "5.2.2. Surface Operations", for the effective creation (in 3D display) and adjustments of a surface.

The `Delete` button destroys the selected surface.

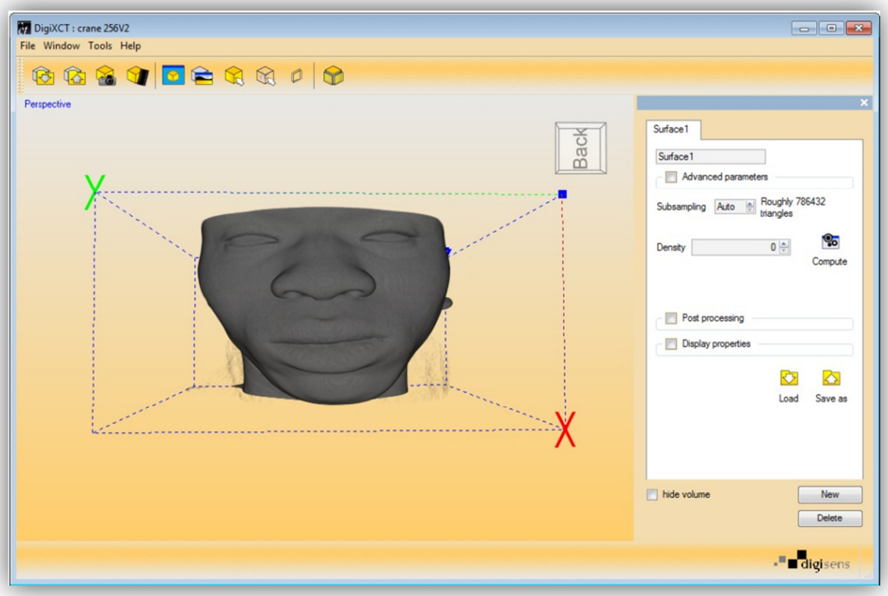


Figure 5.1: Main dialogue box of SRF plug-in

5.2.2 Surface operations

Each created surface corresponds to a tab (figure 5.1). By default, surfaces are named « `surface i` ». Necessary parameters to define a surface are the choice of an isovalue defining iso-density surfaces (`iso-value`), post processings applied onto the surface (`Post Processing`) and display properties (`Display properties`).

Some computations on surfaces may generate objects containing hundreds millions triangles. This triangle amount depends on the chosen iso-value, part complexity and mesh resolution. In order to generate a reasonable amount of triangles, the software proposes a subsampling step increasing the dimensions of an elementary triangle. This parameter is of course informative (but highly recommended because if the necessary amount of RAM needed to compute such mesh is over the limit fixed by the operating system, an application crash may occur) and the user can adjust it.

The button `Compute` computes the corresponding mesh.

5.2.3 Surface operations

Here, we provide details of the accessible functionalities for each surface created. A tab corresponds to each surface created (Figure 5.3).

5.2.3.1 Surface computation

In the example shown in 5.2, the "interesting" density ranges are those between 0.163 and 0.638.

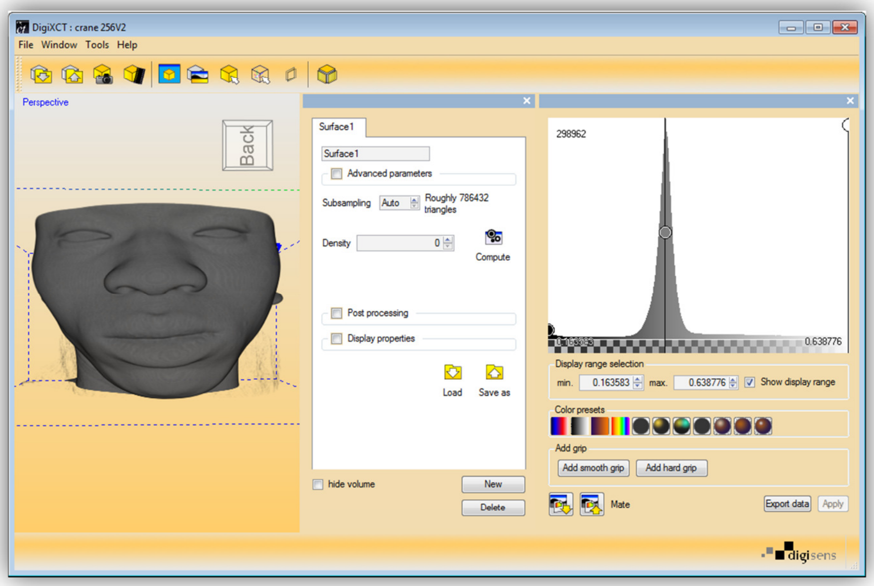


Figure 5.2: Volume visualization of a human head

To extract a surface from this volume, you must:

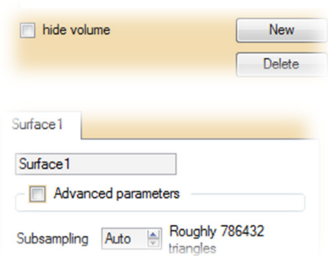
1. Create a new surface (`New` button)

2. In this tab (called *surface 1* by default), enter the density value corresponding to the surface to be reconstructed (`iso-value` at 0.200930 in our example)
3. Click on the `Compute` button to generate the surface.

The `Estimate` button helps to define the density value. The result of this estimation is given in the console (`Window` → `Console`).

Thus you obtain a surface corresponding to the density (grey level of the histogram) of 0.024 in the 3D display.

By default, the surface and the volume are superimposed. The option `Hide` allows you to hide the voxel model.



The information on the right of the name of the surface indicates the number of triangles used to define the mesh. In this example we get 786 432 triangles.

The name of a surface can be changes at any time.

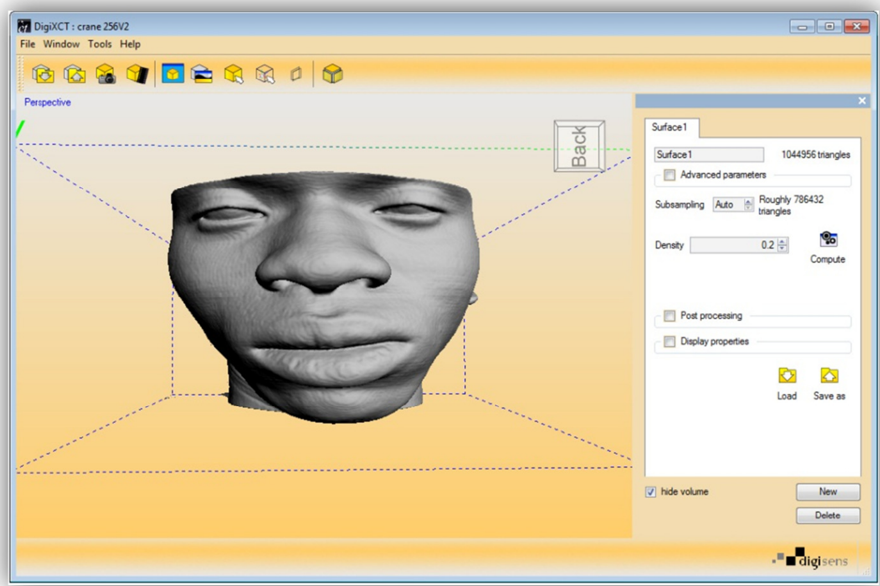


Figure 5.3: Mesh extraction from the human head using an iso-value of 0.2.

5.2.3.2 Post processing

Post processing (`Post Processing`) such as smoothing and/or triangle decimation can be applied to extracted surface (Figure 5.4: Post Processing).

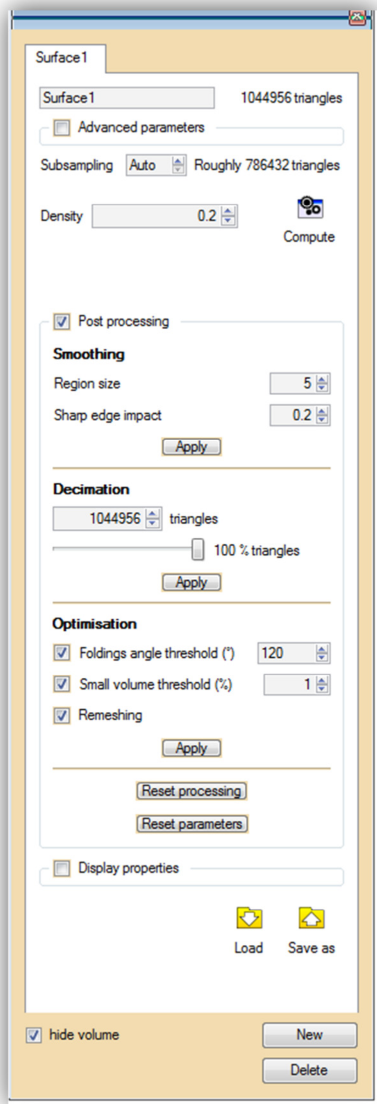
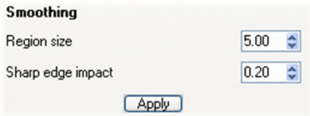


Figure 5.4: Post Processing window

5.2.3.2.1 Smoothing



Two parameters allow the user to adjust mesh smoothing (`smoothing`): `Region Size` and `Sharp edge Impact`. Smoothing allows the user to re-arrange triangles in the mesh defining the surface.

`Region Size` is the window size in which triangles are re-arranged regarding their neighbors.

`Sharp edge Impact` allows preserving, regarding its value, sharp edges in a surface.

Each time one clicks on `Apply` button processing is applied on the current surface. Thus, you can apply recursive post-processing.

Following figures illustrate smoothing:

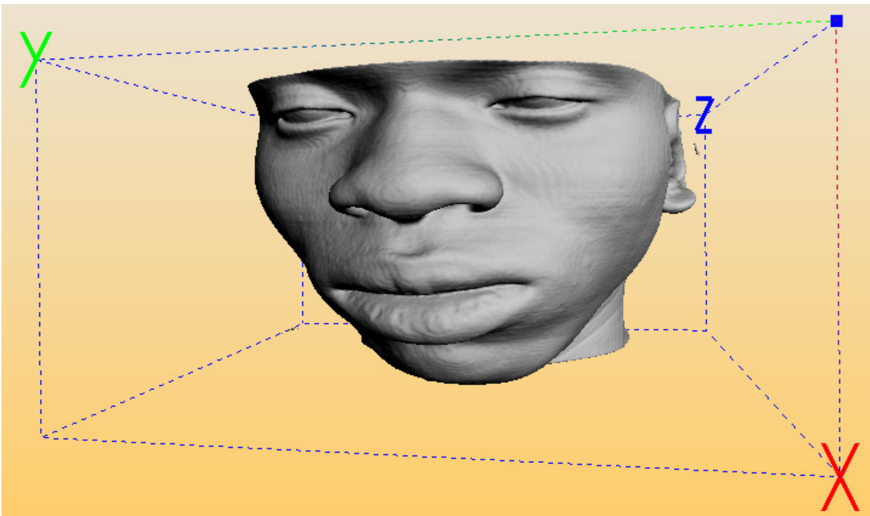


Figure 5.5: Without smoothing

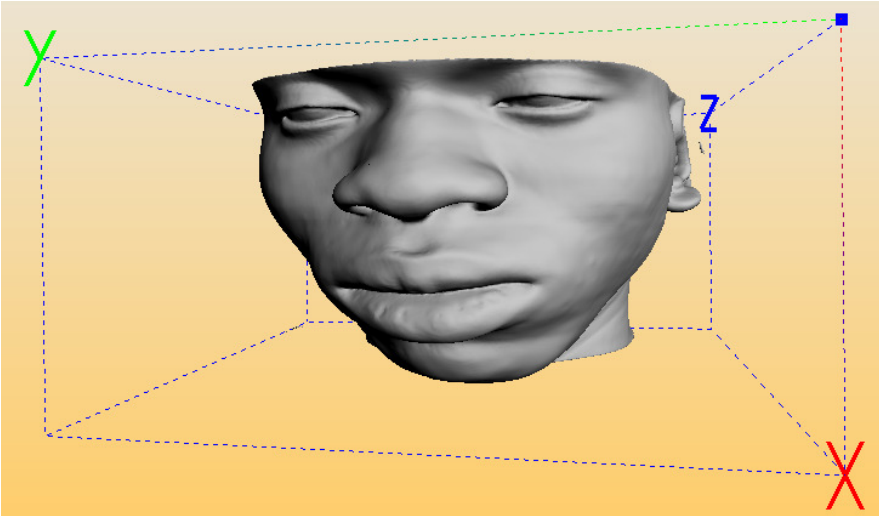


Figure 5.6: With smoothing (region Size = 5.00 and Sharp edge impact = 0.2)

5.2.3.2.2 Decimation

Decimation reduces the number of triangles defining a mesh. One can either fixe the desired number of triangles, or fixing a percentage of decimation regarding the initial number of triangles generated.

`Apply` launches decimation process on the current mesh.

Figures 5.7 and 5.8 show decimation result.

The two post processing can be applied simultaneously.

`Reset processing` button allows the restoration of the initial mesh removing all post processing applied. Those processing will be lost.

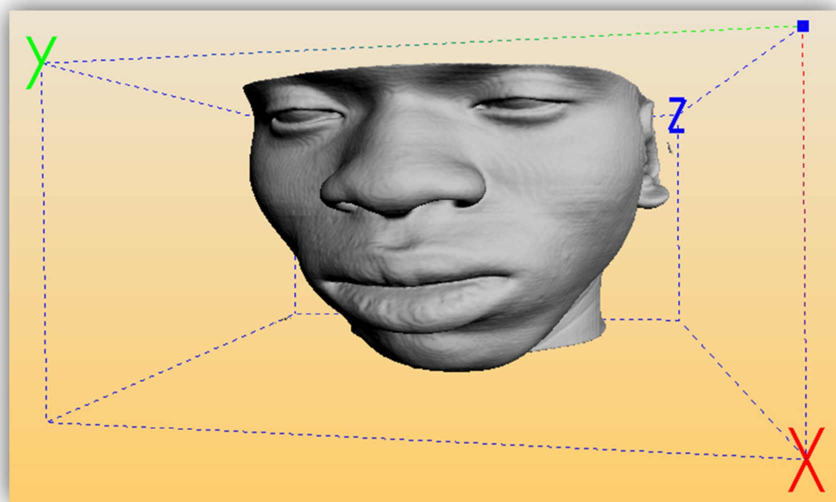


Figure 5.7: Initial surface (1044956 triangles)

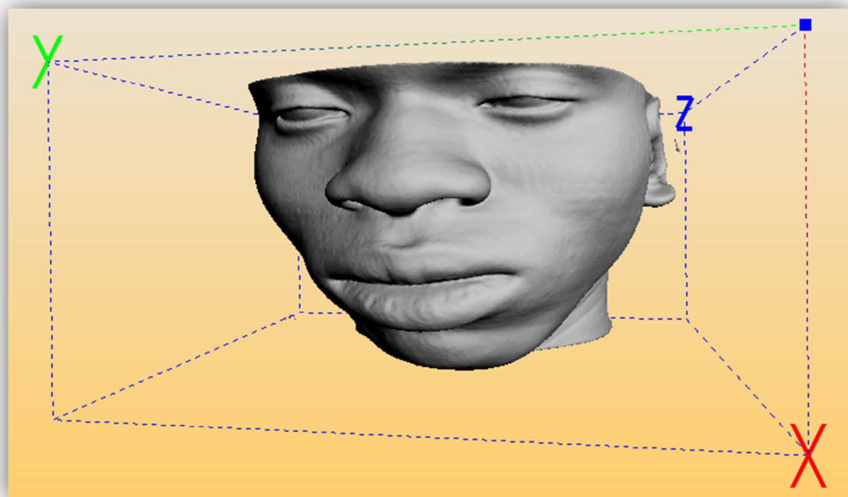


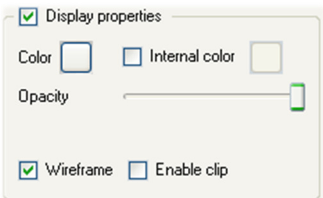
Figure 5.8: Surface with 50% less than initial surface (522478 triangles)

5.2.3.2.3 Optimization

Mesh optimizations are the following:

- `Folding's Angle Threshold`: allows removing neighboring triangles for which the angle between their normal vectors is equal or higher than the defined parameter. This option allows for instance to remove folds in meshes.
- `Small Volume Threshold`: allows removing meshes delimiting a volume lesser or equal to the defined parameter.
- `Remeshing`: This option allows you to perform a remeshing in order to get triangles shape close to equilateral triangles. This computation may be very long regarding the number of triangles to process. A rough estimation of the processing time is indicated.

5.2.3.3 Display properties



(Figure 5.9)

Display properties allow the user to choose the external color of the mesh (`color`) as well as its internal color (`Internal Color`). It also allows defining transparency of the surface (Opacity), to display it using a wireframe rendering, to apply clipping plane onto it (Enable Clip).

Note: clipping plane is independant for each extracted surface.

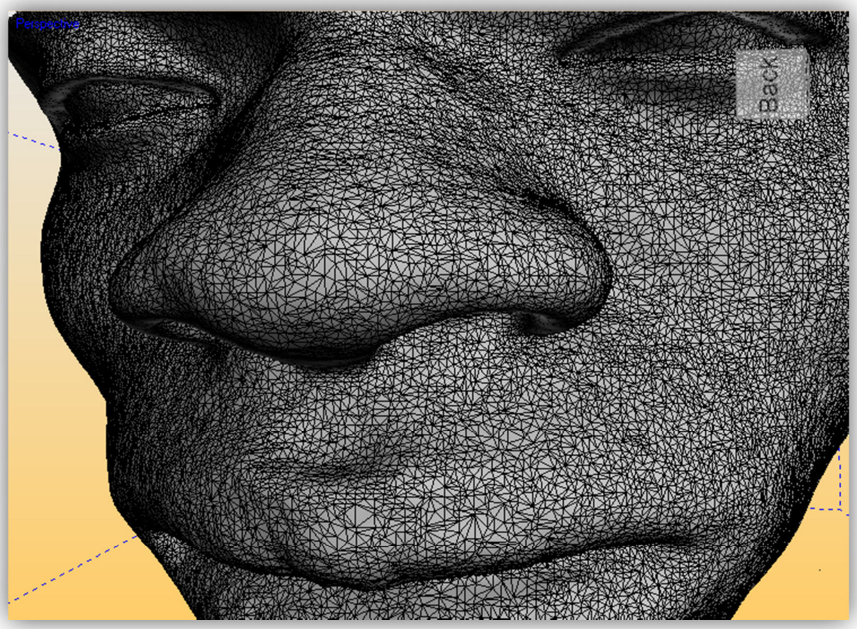


Figure 5.9: wireframe display of our example

5.2.3.4 Advanced parameters

Advanced parameters propose the choice between two mesh extraction algorithms: `Dual Contouring` or `Hierarchical` algorithm.

`Hierarchical` approach is an optimization of `Dual Contouring`. In some cases, it can reduce the number of triangles in a mesh.

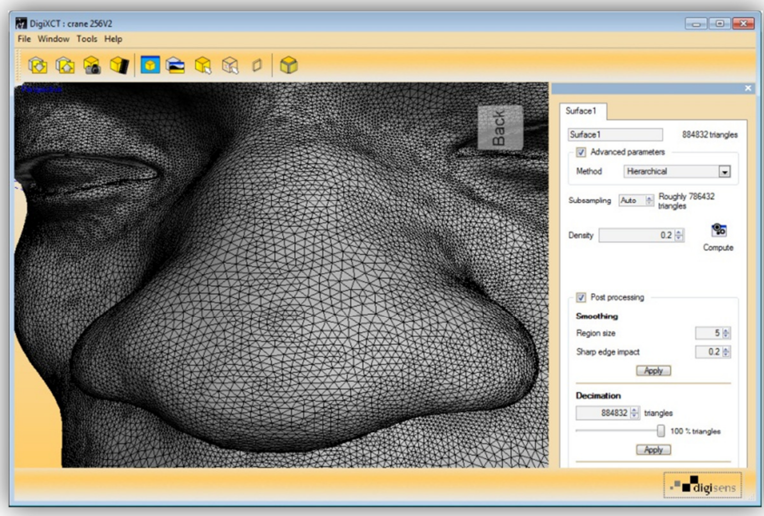


Figure 5.10: Mesh extraction using hierarchical approach

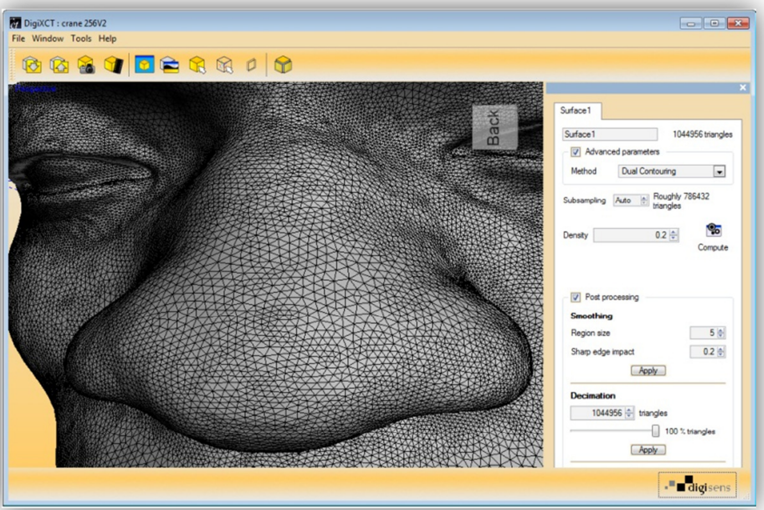


Figure 5.11: Mesh extraction using Dual Contouring approach

5.2.4 Simultaneous visualization of multiple surfaces

It is possible to compute several meshes on a same volume by changing iso-value. For the example presented in this chapter, we can compute a second surface with the iso-value 0.390 (figure 5.12).

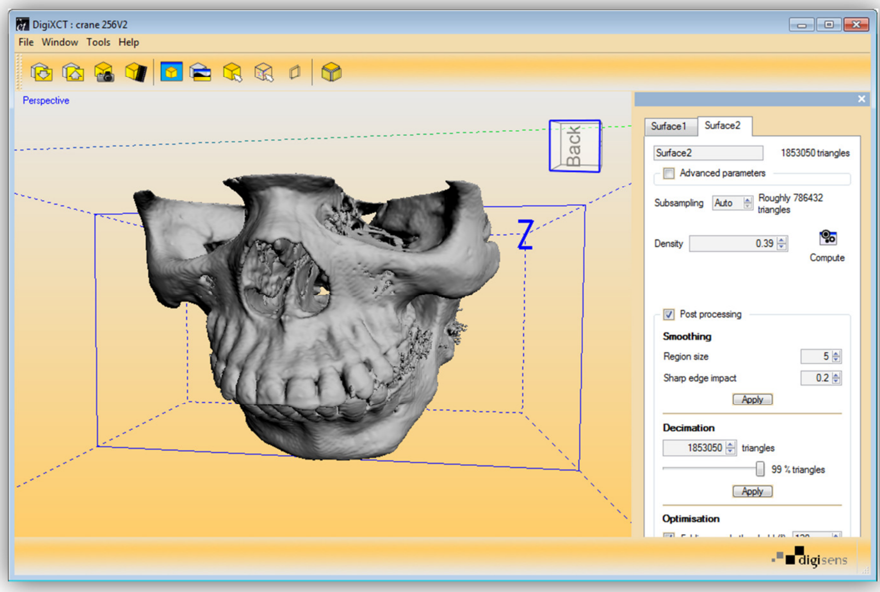


Figure 5.12: Second surface extraction

It is possible to visualize surfaces simultaneously using transparency parameter or/and using clipping plane (Figure 5.13).

5.3 *Saving/ reloading*

Each tab linked to a surface is in association with two buttons `Load…` and `Save…` allowing to save and reload a surface.

Possible file formats are available in read/write mode are:

1. Internal format *.surf*
2. OBJ Wavefront with extension *.obj*
3. STL (ascii) with extension *.stl*

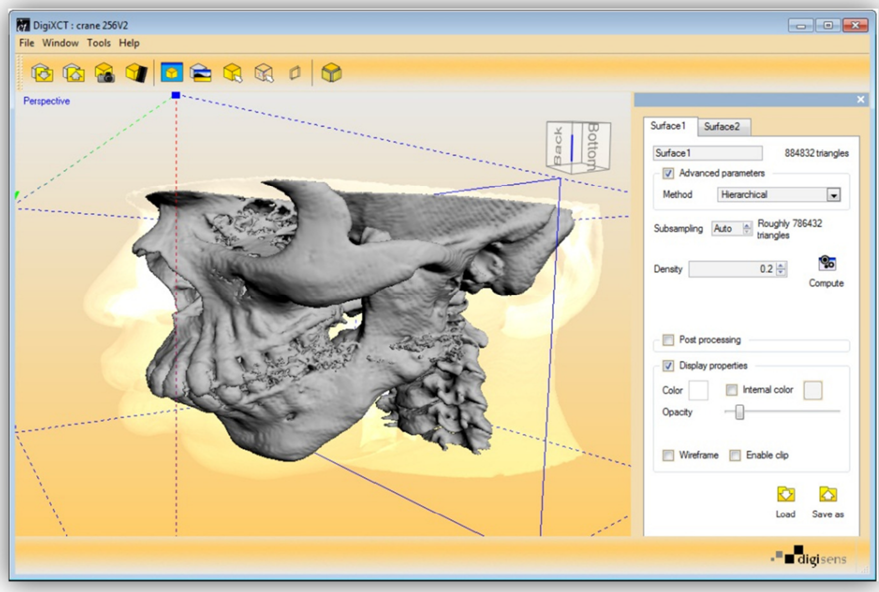


Figure 5.13: Visualization of two surfaces changing transparency

6 Import of tomography slices: DigiSIM plug-in

As with all the plug-ins in the DigiXCT software suite, DigiSIM is protected by a software key.

6.1 General presentation of DigiSIM

This plug-in allows existing tomography slices to be imported, parameters for sizes and corresponding resolutions to be set and the slices for a 3D volume visualization to be “compiled”. When DigiSIM is loaded by DigiCT, the  icon appears in the plug-ins’ icon bar.

6.2 DigiSIM functionalities

To launch the DigiSIM plug-in, you need to launch the command on the `[File → Import Slices…]` menu or click on the  icon to make the import dialogue box appear (Figure 5.1).

The `Input slices` zone allows slice files to be specified: the  button displays a dialogue box to select one of the slice files. The list of slice files is then displayed in the entry box on the left of this button and a window displays the first image from this list. From then on, it is possible to scroll the slices via the cursor (Figure 5.1). It is possible to select an area within the images by specifying `umin`, `umax`, `vmin` and `vmax`, and a special group of slices by specifying `wmin` and `wmax`. `Input resolution` then allows the resolution of the input images to be entered.

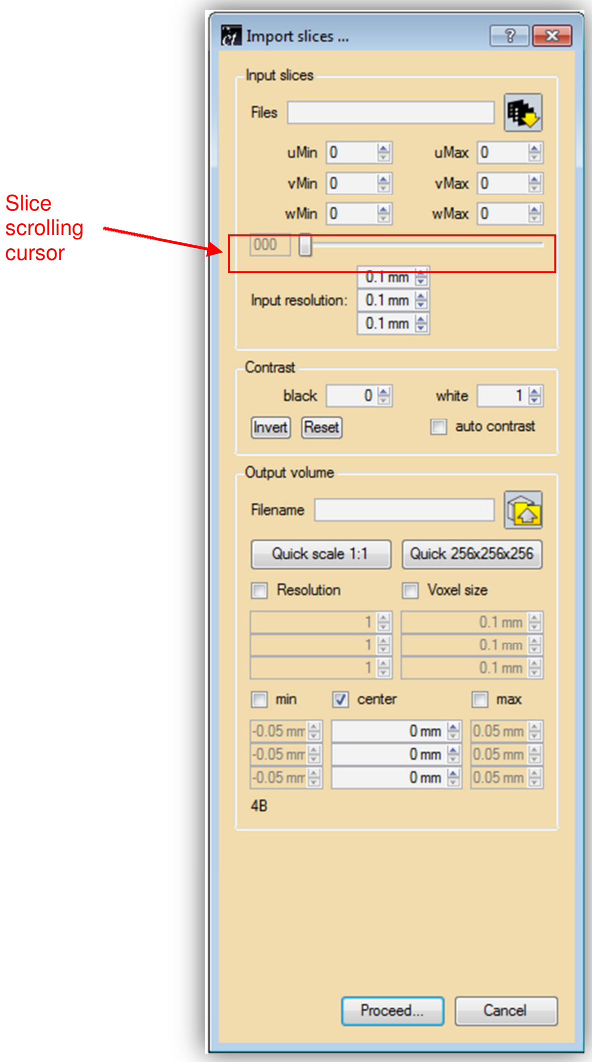


Figure 6.1: Slice import dialogue box

The `Contrast` zone allows the level of black and white to be modified (`black` and `white` labels) or the images to be inversed via the `Invert` button.

The `Output volume` zone allows the name of the output file to be specified:

the  button displays a dialogue box to choose the placement of the volume and its name. An existing file can be overwritten by selecting the header file `header.vol`.

The min point, the max point and the center of the volume can potentially be modified with the actual dimensions of the stack of images. You can also specify the resolution of a volume (256x256x256 by default) and the size of a voxel in mm.

The `Quick scale` buttons allows volume parameters to be set more quickly, either in full resolution or in a lower quality resolution. The import is achieved by clicking on “`Proceed`”.

Once the calculation is finished, a volume file `*.vol` is created in the pre-selected directory. It is possible, from this new volume file, to use all of the other 3D treatment plug-ins, and to export 2D slices in other directions.

Note: the imported slice name has to be numbered with the same number of digits. For example, if the last slice number is `slice_532.tif`, the first slices have to be numbered `slice_0000.tif`, `slice_0001.tif`…

7 Automation of reconstruction tasks: Batch Mode

As with all the plug-ins in the DigiXCT software suite, the batch mode is protected by a software key.

7.1 General presentation

DigiXCT offers the user the possibility of carrying out a group of tasks automatically and without supervision. Batch mode consists of writing a small command script which requests the execution of basic tasks such as reconstruction, display or even other functions.

7.2 Batch mode functioning

To access batch mode, you only need to launch the command of the `[Window → Script]` menu represented by the  icon. The window shown in figure 7.1 appears. The list of basic functions can be obtained by hitting help in the script window then clicking on the `Go` button.

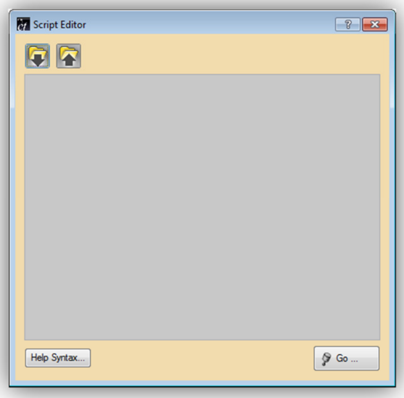


Figure 7.1: Script window

Figure 7.2 shows a script example which allows:

- Loading a calibration file

- Requesting a reconstruction with interpolation of the voxels
- Set the final resolution at 256x256x256
- Launch the reconstruction via DigiFDK
- Display the volume in DigiObs

Execution occurs by clicking on the `Go` button.

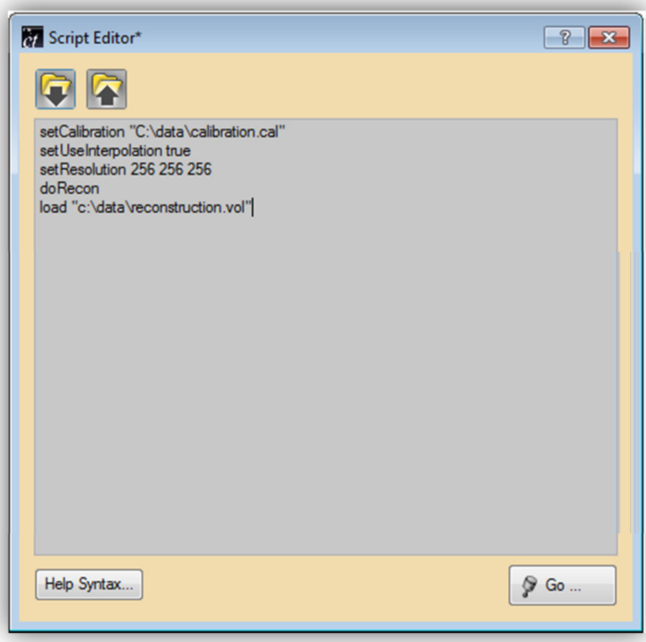


Figure 7.2: Example of script program